METAL

MACHINING

METAL MACHINING

by

LAWRENCE E. DOYLE, Reg. Prof. Engr.
Associate Professor of Mechanical Engineering
University of Illinois

PRENTICE-HALL, INC.
Englewood Cliffs, N. J.

 Library of Congress Catalog Card Number: 53-6818

Current printing (last digit):

18 17 16 15 14 13 12

PRINTED IN THE UNITED STATES OF AMERICA

57798-C

To Hazel

PREFACE

THE PURPOSE OF THIS BOOK is to acquaint the engineer or the prospective engineer with the machining area of production processing from the engineering viewpoint. This engineering approach has two facets. The first explains the reasons for the practices, methods, tools, and machines found in industry. This phase is the basis for the descriptions of metal machining processes throughout the book. The second makes use of principles to adapt practices, arrange methods, and select tools and machines to produce efficiently. This aspect, known as "process planning," is the medium through which the engineer applies a knowledge of the facts and principles of metal machining. The fundamentals of process planning are presented here to add meaning to the study of metal machining and to give the reader exercises in making use of what he learns.

The author is indebted to Professor George F. Schrader for his thorough study and thesis on the subject of "Basic Principles to be Used in Planning for Metal Machining Operations." His work has provided a sound basis for a large part of this book. Professor Schrader has also contributed many valuable comments and suggestions in its preparation.

The author also wishes to acknowledge the contribution of Professor George W. Harper in preparing the material on safety.

To the many manufacturers of machines and tools who have contributed illustrations and information to make this book possible, credit is extended throughout the work.

LAWRENCE E. DOYLE

Urbana, Illinois

CONTENTS

METAL

MACHINING

Chapter 1

INTRODUCTION

THE REAL PURPOSE OF ENGINEERING is to create useful goods and services, to make them better, cheaper, and more abundant. The physical conveniences of our everyday lives are constant evidence of engineers' efforts. Swift and comfortable transportation, instant and widespread communication, efficient appliances, and light and power from electricity are only a few of the conveniences available to us because of the ceaseless endeavor of engineers to transmute ideas into realities. And one of the most important aids to engineers in realizing their goals is metal machining, the cutting or shaping of metals by power-driven machine tools.

The importance of metal machining. Metal machining makes most engineering projects possible. It is the basis for all manufacturing—directly in fabricating metal products like sewing machines, stoves, automobiles, and electric motors; indirectly in creating the machines that produce goods like paper, drugs, food, furniture, and clothes. Engineers work with many kinds of materials, but metals are the most important because of their strength, durability, and stability. Even if metal does not appear in the final product, it often plays a leading role in the fabrication of goods. Plastic parts are formed in steel dies; wood is cut with metal tools; food products are packaged by machines of metal and often packed in metal cans. Wherever metal is utilized, we can be sure it reaches the stage of usefulness through machine tools and the work they do. Machine tools have been called "the machines that make the machines that make the machines." To that description might be added, "that make all the products of our industrial era."

Many specialists contribute to the outpouring of industry. Research brings forth new ideas and clarifies old ones. The research engineer often uses elaborate and intricate equipment that is

machined from metal. Much of that equipment he must improvise and design himself. An understanding of metal machining can certainly help the research engineer get the most out of his equipment with the least effort.

Design engineers develop ideas into workable devices. In preparing to produce an automobile, for example, mechanical engineers versed in thermodynamics and combustion design an engine to convert fuel into power efficiently and smoothly. Others proportion structural and drive members for strength, durability, and comfort. Electrical engineers are needed for the ignition, lighting, and starting system. Stylists bring out pleasing lines for the exterior and striking appointments for the interior of the body. Before production can be started, models must be made and tested. The design engineers are expected to create a vehicle that performs as near to perfection as possible and that stands up for a long time. But the automobile must also be designed so that it can be made and sold at a low price. That is true of all manufactured articles that have to meet competition. Designers must be acquainted with production methods, particularly with metal machining, if their designs are to be practicable and economical to make.

After a product design has been approved and released, the production engineer is responsible for furnishing the plans for production and the physical means for executing them. The plans include the routings that specify the operations that must be done on each part. Other plans involve the scheduling of materials and men to assure a reliable flow of products. Machine tools, cutting tools, jigs, fixtures, dies, gages, etc. must be devised, procured, and made ready as needed. The production engineer must have an intimate knowledge of metal machining to carry out these functions.

The advantages of metal machining. Objects can be made from metal by a number of processes. Among them are founding, forging, forming, rolling, welding, and cutting. Metal machining includes the cutting processes. An article can often be made by any of several processes. Sometimes the most convenient method is chosen, but normally that process is selected that produces the required results at the lowest cost.

A method may call for inexpensive equipment but require that much time be spent on each piece. Such a method is often the most

economical to make only a few pieces. As an example, a part can be cut from standard bar stock on general-purpose machine tools at a cost of $10 for each piece. The cost for 5 pieces is $50, and for 25 pieces is $250. If the part is cast, a pattern must be made at a cost of $25. Molds can be made and pieces cast for $2 each, and the machining charge to finish the castings is $5 each. To make 5 pieces by casting and machining results in a total cost of $60; for 25 pieces, $200. Obviously machining from bar stock is preferable for 5 pieces, but the making of castings and machining them is cheaper for 25 pieces. If the quantity to be manufactured is very large, automatic machinery entailing a large investment may be utilized to realize a low net cost for each piece produced.

Metal machining is advantageous in many cases. The basic machine tools are versatile, and almost any piece can be made with them. Large amounts of material can be removed by machining when necessary, and pieces can be cut off parent material or separated from each other without excessive waste. Parts can be machined from standard shapes like bars and plates.

Machining is not limited to making parts in small quantities. It has advantages for large- as well as small-quantity production. Surfaces can be machined to almost any degree of accuracy and truth. In fact, the most accurate surfaces can be obtained at reasonable cost only by machining methods. Parts that are formed roughly by other processes, like founding and forging, normally have some or all of their surfaces refined by machining. For instance, most engine blocks are cast, and then their cylinders, faces, and bearing surfaces are machined. Certain machining processes, like grinding, are capable of finishing the surfaces of very hard substances.

The meaning of metal machining. In all metal cutting operations an edged tool is driven through material to separate chips from the parent body. All else that occurs merely contributes to that action. Metal may be cut by simple hand tools such as hammer and chisel, file, saw, or stone. These are used today to remove metal otherwise inaccessible or in small amounts. At one time such tools were about the only means available for cutting metals. Obviously the articles cut from metal solely by hand tools were few and crude. Such methods are slow and laborious and require great skill to guide the tools to produce true surfaces.

With the advent of the industrial revolution, the invention and development of devices like the steam engine and textile machinery called for faster and more accurate methods of cutting and forming metals. Machines were devised to apply power to metal cutting and hold and move workpieces surely and precisely. These superior tools were given the name of *machine tools,* in contrast to hand tools, and the work done by them was called *metal machining.* The planing, turning, drilling, and boring machines came into being early. At first it was considered quite an accomplishment just to make a few articles of metal; later the demand arose for quantity production. Machining methods were applied to firearms and clocks and to new inventions, like the reaper and the sewing machine. Other machine tools like the milling machine, turret lathe, and grinding machine were developed to cut metal faster, reduce labor, and bring about greater precision. To meet the demands of the present century for production in very large quantities, highly specialized and automatic machine tools have been developed. Up to the present time the improvement of machine tools and machining methods has gone steadily forward and gives no sign of faltering.

Some factors in metal machining. It has been said that the loss of a few ounces of metal is enough to make useless an automobile engine, which weighs several hundred pounds. Those few ounces of critical material are on the finished surfaces of the mating parts that make the engine a functioning mechanism. In a good engine, those surfaces must have definite shapes and features. The purpose of all machining is to finish surfaces as required.

The kind of surface produced in a metal cutting operation depends upon the shape of the tool and the path it traverses through the material. If a workpiece is rotated about an axis and a tool is traversed in a definite path relative to the axis, a surface of revolution is generated. If the tool path is parallel to the axis, the surface is a cylinder, as indicated by Fig. 1-1 A. That is called *straight turning* or just *turning.* An inside cylindrical surface is generated in the same way by *boring,* as depicted in Fig. 1-1 B. If the tool path is straight but not parallel to the workpiece axis, a conical surface is generated. That is called *taper turning,* as in Fig. 1-1 C. Both outside and inside tapers can be generated. If the tool is directed in a curved path, as shown in Fig. 1-1 D, a profile of varying diameter

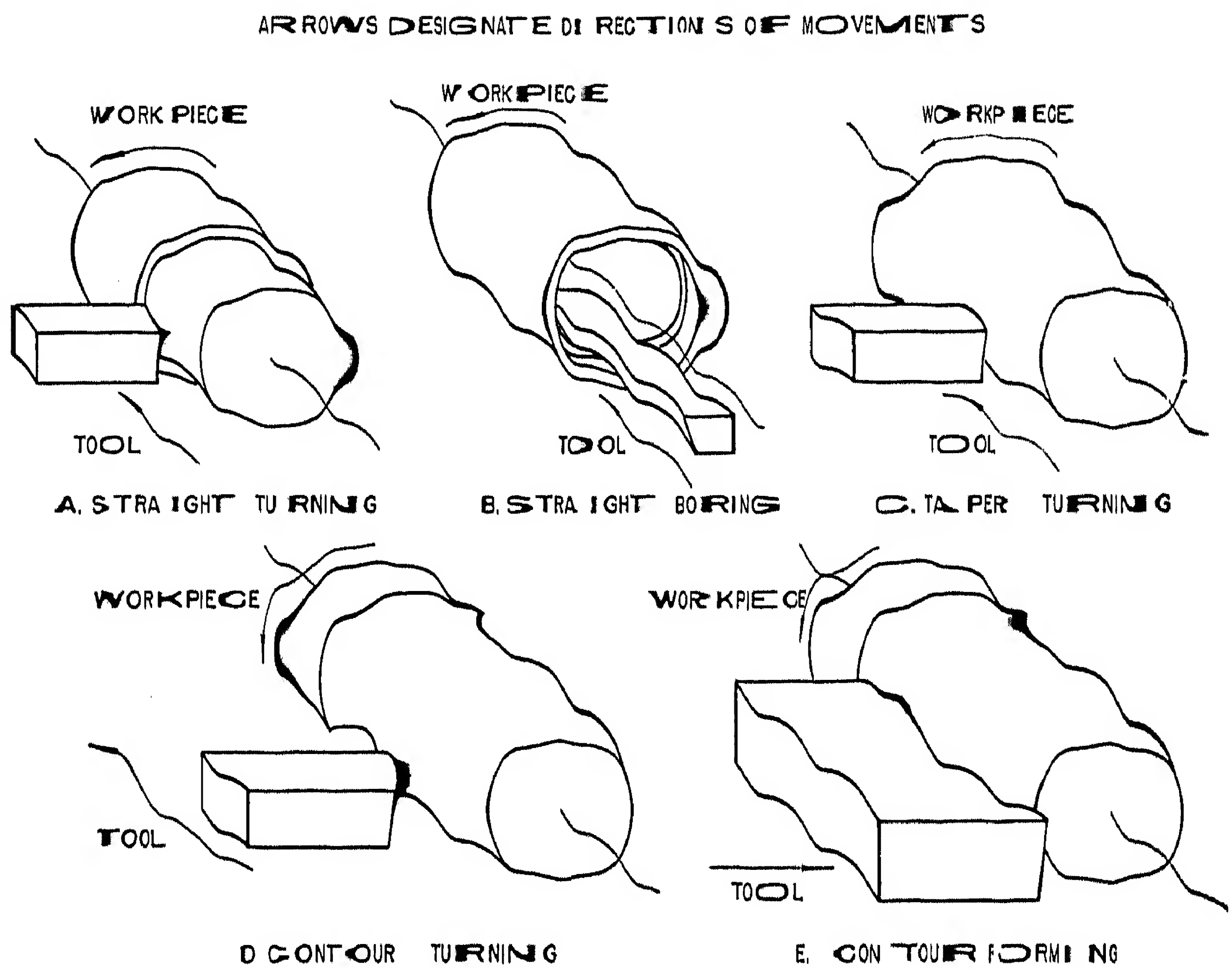


Fig. 1-1. Diagrams showing how surfaces of revolution are generated and formed.

is generated by *contour turning*. In the foregoing examples, the shape of the surface generated depends more upon the path than the form of the tool. A surface of revolution may also be machined by plunging a tool into a revolving workpiece. The profile cut in that way corresponds to the form of the cutting edge of the tool. *Contour forming* done in that way is illustrated in Fig. 1-1E. Straight and tapered surfaces may be formed in a similar manner.

A plane surface on the end or shoulder of a workpiece may be generated by revolving the piece and feeding a tool at a right angle to the axis as shown in Fig. 1-2A. That is called *facing*. Planes may also be generated by a series of straight cuts, without revolution of the workpiece, as illustrated in Fig. 1-2B. If the tool is reciprocated and the workpiece is moved a crosswise increment at each stroke, the operation is called *shaping*. *Planing* is done by reciprocating the workpiece and moving the tool a little for each stroke. Irregular

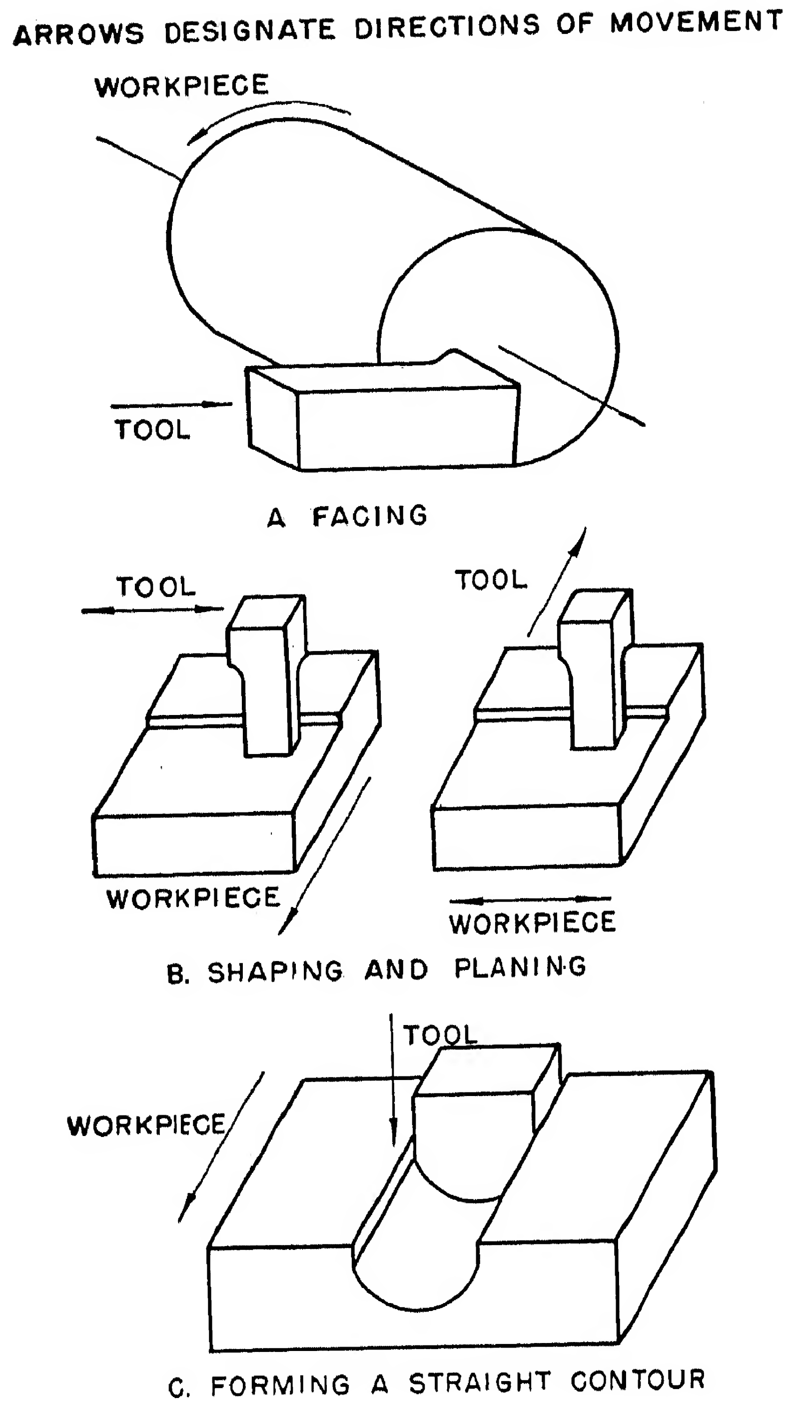


Fig. 1-2. Diagrams showing how plane surfaces are generated and formed.

contours can be machined by these methods by varying the depth of cut or by using a formed tool as indicated by Fig. 1-2 C.

Surfaces may be machined by tools having a number of edges that cut successively through the material. Drills for opening holes are of this type. A drill may turn and be fed into the workpiece, or the piece may revolve while the drill is fed into it. Boring also is often done with tools having several edges. Plane and contoured

surfaces are machined by milling cutters. A milling cutter has a number of teeth on its periphery. The teeth remove chips as the cutter revolves and moves over the surface. Typical milling cuts are shown in Fig. 1-3.

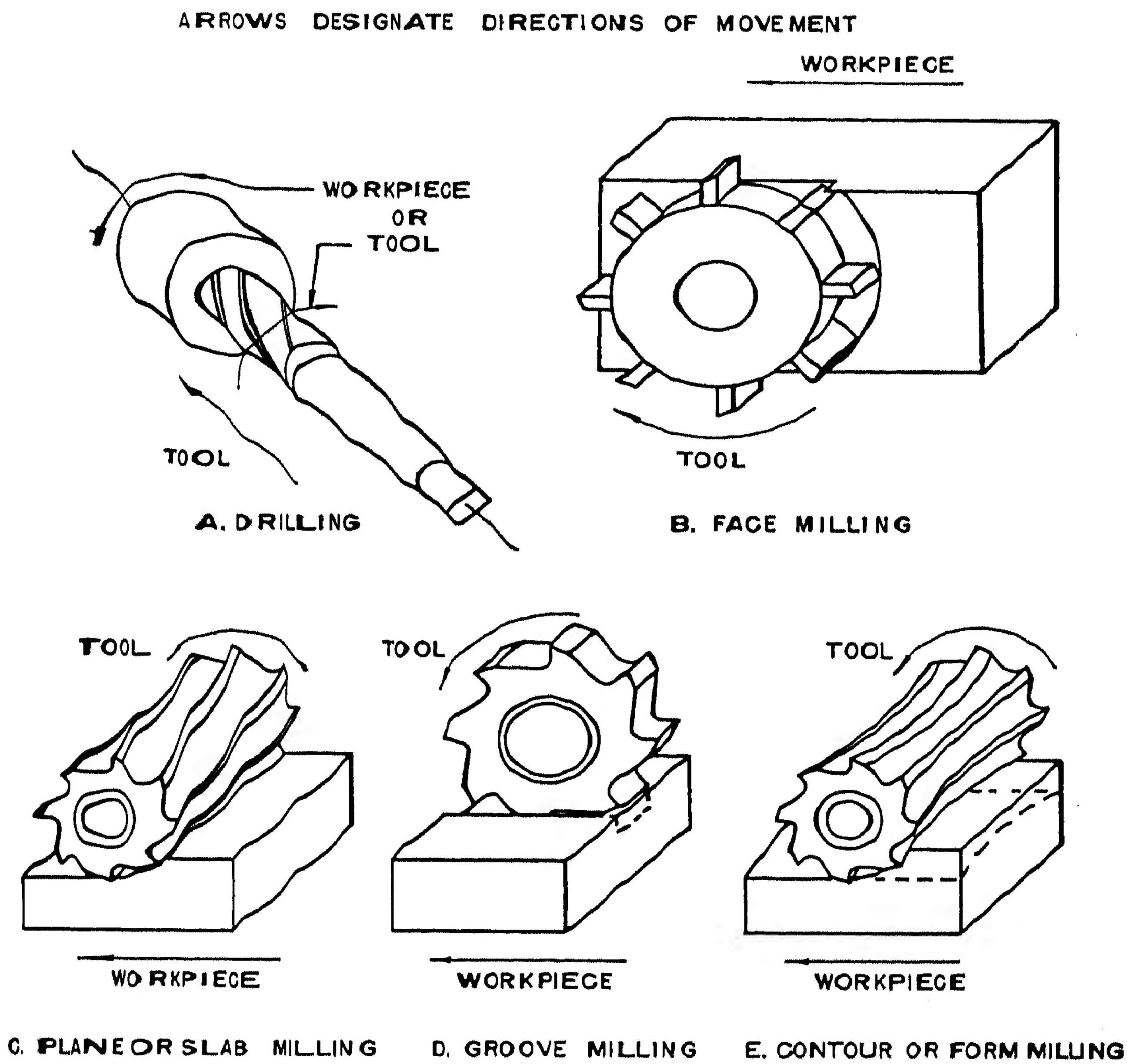


Fig. 1-3. Typical surfaces cut by multi point tools.

The examples that have been given illustrate that metal machining is called upon to produce many kinds of surfaces. Some are simple geometric surfaces, others quite complex. Most workpieces have several surfaces that must be finished. In addition, some surfaces must be quite accurate, others need not be so. Some surfaces must

be smooth, other can be rough. Satisfying all these conditions calls for a large variety of arrangements.

In addition to the variety of surfaces that must be machined, other factors add to the number and variety of situations that must be met in metal machining. Some workpieces are small, others are heavy and bulky and cannot be manipulated easily. Sometimes there are many different parts, but only a few pieces of each kind are required. In that case the equipment and tools must be versatile. Sometimes a large number of identical parts must be produced and the equipment must be able to duplicate them automatically and rapidly. Some materials are easy to cut, others hard. It is conceivable that a machine tool with suitable accessories could be devised to satisfy all the conditions under which metal machining must be done. However, such a machine would be quite intricate and costly, would require considerable skill for its operation, and has not been found feasible.

The evolution of machine tools has culminated for the most part in a species of relatively simple machines. Each type of machine tool is most efficient for certain kinds of work. Lathes give best results in machining surfaces of revolution on small or medium-size parts. The same operations on large and bulky workpieces are more conveniently done on a vertical boring machine. Small pieces may have flat surfaces machined on a shaper, large pieces on a planer, if it is advantageous to use single point tools. The same work may be done on a milling machine where a multitooth cutter is justified for more rapid production. Generally, each type of machine tool makes use of certain kinds of cutters. Single point tools ordinarily are associated with lathes, shapers, and planers, although not exclusively. Milling machines usually rotate cutters with multiple teeth. Grinding machines are designed to utilize rapidly rotating abrasive wheels.

The analysis of metal machining operations. The subject of metal machining is not hard to understand but does require that many facts be learned about types of tools, machines, and auxiliary equipment, their arrangements, capabilities, and limitations. But a collection of facts by itself does not constitute an understanding of the subject. The reasons behind these facts, why tools, machines, and procedures are as they are, must be understood to give them

meaning. Also, it is essential to know how to reason from these facts to solve the problems that metal machining presents. A study of the subject of metal machining is of most value to the engineer if it teaches him how to make logical selections, arrangements, and analyses of tools, machines, and equipment to produce articles of metal efficiently. That approach is emphasized in this text.

Questions

1. Why is metal machining important to engineering?
2. What are some of the advantages of metal machining as compared to other processes for making objects?
3. Describe the basic functions of (a) turning, (b) boring, (c) facing, (d) drilling, (e) shaping, (f) planing, and (g) milling.
4. Why are there different kinds of machine tools?

Chapter 2

DIMENSIONS AND TOLERANCES

DIMENSIONS SPECIFY THE IDEAL size, shape, and other features of a piece. Tolerances designate how much the piece may differ from perfection.

Interchangeable manufacture. Objects are machined to specific sizes so that they will perform certain functions. Interchangeable manufacture is based on the idea of making a machine part close to a definite size and shape so that the piece will fit readily into place and function properly. Examples of devices that depend upon interchangeable manufacture are bicycles, automobiles, washing machines, watches, and cameras. A timing gear for an automobile, for instance, can be taken at random and put in place, and it will function properly on the model for which it is designed.

Interchangeability provides a number of benefits in the manufacture of mechanical devices. In production, all pieces of one kind can be treated in the same way because they are alike. In assembly, time is saved because no fitting is needed. For the user, repairs are simplified because worn parts can be replaced easily. Interchangeable products can be standardized, like nuts and bolts. With interchangeable manufacture, some parts can be made in one plant and others in another, with the assurance that they will fit together.

Without interchangeable manufacture, parts must be fitted or selected when assembled. In fitting, a part may be machined by a cut and try process until its fits a mating part. That may be satisfactory if only one assembly is to be made. Pieces may have to be filed or scraped at assembly to make them fit. That is slow and expensive. For selective assembly, parts may be segregated into groups corresponding to various size ranges. A piece is selected from a group to match the group from which its mating part is taken.

Basic concepts. A dimension is usually thought of as a length between two points, lines, or planes. Such dimensions have to be considered for all objects that are machined, but other features also have the properties of dimensions and often must be taken into account, especially for precision work. They are the geometric relationships of surfaces and surface quality. Their significance as dimensions will be included in this chapter.

When a certain dimension is specified, it is not expected to be realized exactly. If a round piece is to be made with a 2 in. diameter, that does not mean that it will have a finished diameter of exactly two inches. If enough time is spent on the task, and the equipment is available to prove the results, the diameter can be made very close to two inches. Whether one or a number of pieces is to be made, the closer that each piece must be made to a stated size, the higher the cost. Since an exact dimension cannot be realized, some error in meeting it must be accepted. The amount of variation permitted in a dimension is called *tolerance*.

Tolerance must not be confused with *allowance* which is the intentional difference between the corresponding dimensions of two mating parts. Allowance is the least clearance or most interference between mating surfaces. In Fig. 2-1, the smallest hole size is 1.000 in. The largest diameter permitted for the small end of the shaft is 0.999 in. The least clearance between the shaft and small hole is 0.001 in. That is the allowance for the running fit. The minimum diameter of the large hole is 1.500 in., and the largest shaft diameter is 1.5015 in. The most interference of the force fit is 0.0015 in., and that is its allowance.

Other terms appear in discussions of dimensions and should be understood. *Basic size* is the exact theoretical size from which variations are made. In Fig. 2-1, the basic size of the large hole is 1.500 in., and of the small hole 1.000 in. A *standard size* is a recognized or accepted size corresponding to subdivisions of a unit of length. The common fractions of an inch are called standard sizes. *Nominal size* refers to the standard size that approximates a dimension. In Fig. 2-1, the large end of the shaft has a nominal size of 1½ in. The range in which a dimension is permitted to vary is said to have *limits* at its ends. A one-inch dimension that may

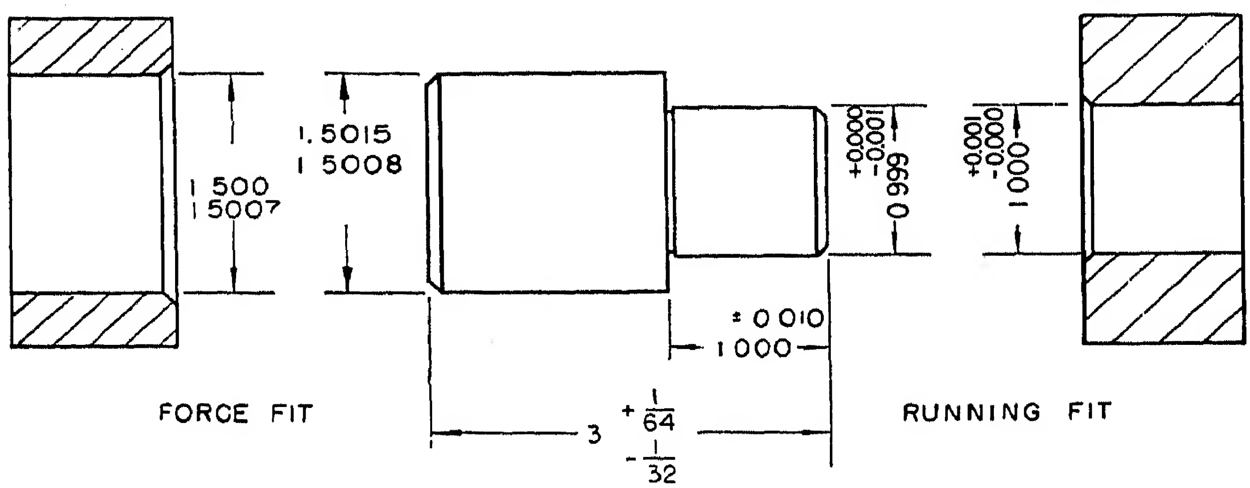


Fig. 2-1. Typical dimensions and fits.

vary 0.010 in. either way from its basic size is confined by the limits of 1.010 and 0.990 in.

Linear Dimensions

Ways of writing tolerances. Tolerances may be written in several ways for linear dimensions. In the *bilateral* form, a dimension is permitted to vary in two directions. The $3\,^{+1/64}_{-1/32}$ in. and 1.000 ± 0.010 in. dimensions of Fig. 2-1 are bilateral. A *unilateral* tolerance is applied all in one direction. That form is illustrated by the $0.999^{+0.000}_{-0.001}$ in. and $1.000^{+0.001}_{-0.000}$ in. dimensions of Fig. 2-1. Sometimes only limits are specified, in the form of a *limiting* dimension, like the 1.5015/1.5008 in. and 1.500/1.5007 in. dimensions of Fig. 2-1.

Each way of writing tolerances is advantageous for certain conditions. The bilateral form implies that variation on either side of basic size is acceptable. It is commonly used for dimensions that are not critical. A unilateral tolerance signifies more danger in erring to one side than the other of basic size. In the case of the small hole and shaft of Fig. 2-1, the shaft diameter cannot go beyond its upper limit, nor the hole below its lower limit, without jeopardizing the allowance. Unilateral tolerances are convenient because they reflect what takes place in some machining processes. For instance, a drill or reamer of standard size normally cuts a hole larger than its own size. Limiting dimensions are often used for production where

results are not measured but rather gaged to ascertain that each dimension falls within limits.

Tolerances are often assigned to conform to certain systems of design. Two of these are *standard hole practice* and *standard shaft practice.* Both call for unilateral tolerances. In standard hole practice, also called the *uniform minimum hole system,* the basic size of the hole is set at a standard size, and the basic size of the mating shaft at the standard size less the allowance for a running fit. The basic size of the shaft is the standard size plus the allowance for a tight fit. The tolerance on the hole is plus, and on the shaft minus. The shaft diameters and holes of Fig. 2-1 are dimensioned in accordance with standard hole practice. That system permits the use of standard reamers for finishing holes and standard "go" plug gages. Standard shaft practice prescribes that the high limit of a shaft be the standard size, and the low limit of the hole the standard size plus the allowance for a running fit or minus the allowance for a tight fit. The hole is given a plus tolerance, and the shaft a minus tolerance. This system is convenient for standard commercial shafting because various parts can be mated to it without machining the shaft.

The selection of tolerances. Most dimensions may vary a large amount without harm. That is particularly true of dimensions on a part between surfaces that do not mate with or even come close to surfaces on other parts in a mechanism. The length of a handle on a machine is an example. It usually would make no difference if a handle in the clear were one-half inch longer or shorter than its nominal size. But handles of a certain kind generally do not vary by as much as an inch in length because it is just as easy to make them to a much smaller tolerance. In metal machining, tolerances of 1/32 to 1/16 in., can normally be held as easily as larger tolerances. Consequently, a dimension to be machined that is not important may be given a tolerance of ± 0.010, ± 1/64, or ± 1/32 in., depending upon convention. Quite often a whole or fractional number without an affixed tolerance is written for a dimension. In such cases, custom or a note on the drawing is followed in working to a broad tolerance.

In most devices, some dimensions must have small tolerances. That is generally true of dimensions that determine how mating

parts fit and work together. If the bearings in an automobile engine are loose, the engine is noisy. If a gear does not fit well on its shaft, it will probably not mesh properly with its mating gear nor transmit power smoothly. Journals and bearings must be made with small tolerances so that any two mating parts will fit together snugly. Some functions require smaller tolerances than others. In the manufacture of a mechanism, the smaller the tolerances, the better the initial average performance. Also, small tolerances assure that more metal is provided on wearing surfaces. Thus, the quality of a mechanical product is enhanced by dimensioning its critical surfaces with small tolerances. But small tolerances are costly to hold. Larger tolerances mean a lower production cost but a compromise with quality. What is desirable in every case is tolerances that give a satisfactory level of performance at a reasonable cost.

Fits. Mating parts must fit together in a definite way to perform a definite function. If they are required to move with respect to each other, they must have what is called a *clearance* or *running fit.* If one is to be held tightly by the other, they must be engaged in an *interference* or *press fit.* Although an infinite number of fits is possible, experience has proved that a relatively few fits suffice for most applications. For general purposes, the American Standards Association has proposed eight classes of fits. That proposal is summarized in Fig. 2-2. Detailed working tables of the tolerances

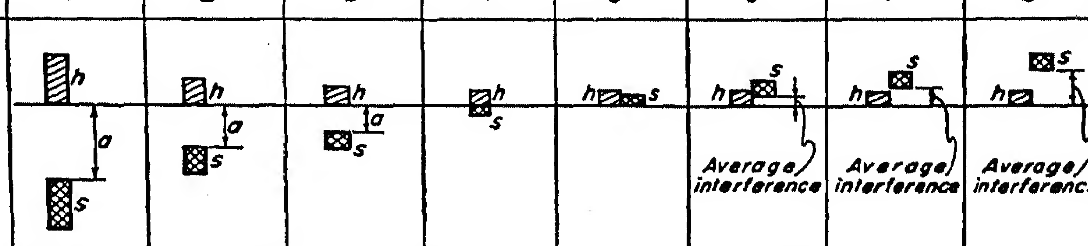
Class of Fit	1	2	3	4	5	6	7	8
Bar Diagram (Basic hole system)								
Hole tolerance	$0.0025\sqrt[3]{d}$	$0.0013\sqrt[3]{d}$	$0.0008\sqrt[3]{d}$	$0.0006\sqrt[3]{d}$	$0.0006\sqrt[3]{d}$	$0.0006\sqrt[3]{d}$	$0.0006\sqrt[3]{d}$	$0.0006\sqrt[3]{d}$
Shaft tolerance	$0.0025\sqrt[3]{d}$	$0.0013\sqrt[3]{d}$	$0.0008\sqrt[3]{d}$	$0.0004\sqrt[3]{d}$	$0.0004\sqrt[3]{d}$	$0.0006\sqrt[3]{d}$	$0.0006\sqrt[3]{d}$	$0.0006\sqrt[3]{d}$
Allowance	$0.0025\sqrt[3]{d^2}$	$0.0014\sqrt[3]{d^2}$	$0.0009\sqrt[3]{d^2}$	0				
Av. interference					0	0.00025d	0.0005d	0.001d

Fig. 2-2. A diagram of standard fits based upon A.S.A.B.4a-1925 classification of cylindrical fits. (**From Spotts,** Design of Machine Elements, **p. 530, Prentice-Hall, Inc., 1948.**)

and allowances for common dimensions in the standard fits are available in standard bulletins and handbooks.

A fit depends upon the tolerances of the mating dimensions and upon the allowance. The larger a dimension, the more difficult it is to hold it to a certain tolerance. That is reflected in the relationship expressed in the chart of Fig. 2-2, where tolerance is proportional to the cube root of diameter.

In Fig. 2-2, Class 1 gives the loosest fit with the most tolerances and is used where some confinement is desirable but extreme accuracy is not essential—agricultural and mining machinery, for example. The liberal allowances and tolerances of Class 2 provide freedom for high speeds and pressures. Class 3 is intended for comparatively low speeds and pressures, for sliding fits in machine tools and automobiles. Class 4 is a snug fit and requires considerable precision. Selective assembly is usually required for Class 5, called a wringing fit. Class 6 is a tight fit for more or less permanent assemblies, such as the fixed ends of studs for gears and pulleys. Considerable pressure is required to assemble parts for Class 7, called a medium-force fit. This is the tightest fit for holes in cast iron and is used for such parts as locomotive and car wheels, dynamo and motor armatures, and crank disks. Class 8 is a heavy force or shrink fit for steel holes.

Classes 1 to 4 are clearance fits; Classes 7 and 8 are interference fits; but Classes 5 and 6 may give either clearance or interference if randomly assembled and are called transitional fits. The particular class of fit for any one application must be selected to suit requirements. Then from the formulas of Fig. 2-2 or from handbook tables, the proper allowance and tolerance can be found for the dimension required.

Geometric Dimensions

The nature of geometric dimensions. Geometric dimensions define the relationships among surfaces on a part. Typical requirements are that certain surfaces must be parallel, that certain surfaces must be square with each other, or that certain diameters must be concentric. Specific angles between intersecting lines or planes may also be considered geometric dimensions.

Geometric dimensions must have tolerances, the same as linear dimensions. For instance, an angle of 45° between two planes may be designated as $45° \pm 2°$, $45° \pm 30'$, $45°^{+5°}_{-0°}$, or whatever the re-

quirements dictate. A common specification for geometric dimensions is in the form of a note on a drawing. Typical notes are:

"Top and bottom surfaces must be parallel within 0.002 in. in 12 in."
"The two faces must be square within 0.001 in. in 12 in."
"Ground diameters must be concentric within 0.003 in. total indicator reading."

Such notes often are supplemented by witness lines and arrows pointing to the lines or surfaces involved. Geometric tolerances are frequently specified in accordance with the methods of checking them. For example, the third note above implies that one of the diameters is to be revolved in contact with a vee block while an indicator bearing on another diameter shows the rise and fall of the second surface as it turns. The note specifies that the indicator pointer must not show more than 0.003 in. change in any one direction while the diameters are rotated through 360°. The maximum indicator reading in such a case is twice the actual amount of displacement between the axes of the diameters.

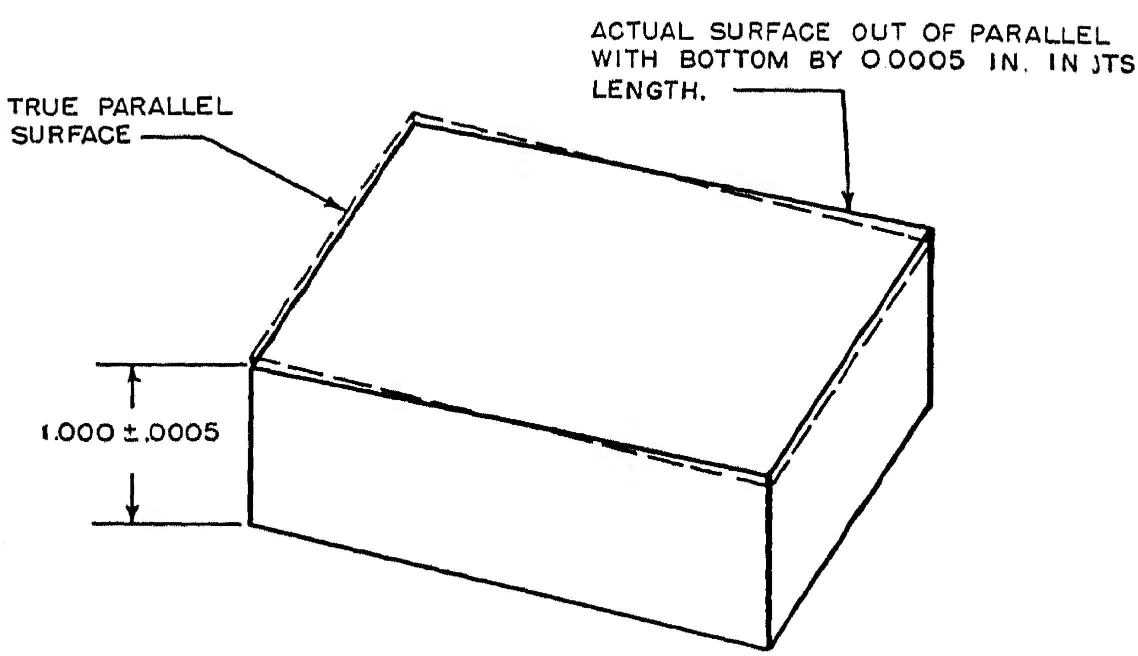


Fig. 2-3. An illustration of the relationship between linear and geometric tolerances.

The effect of geometric tolerances upon linear tolerances. Geometric tolerances often are related to linear tolerances. An illustration is given by Fig. 2-3. The top and bottom of that block

are out of parallel by 0.0005 in., and that in effect takes up half of the tolerance of the thickness of the piece. It is obvious that the top and bottom of the block must be parallel within 0.001 in. if the linear tolerance of the thickness is to be held at all. Situations like this provide strong incentives to keep variations in parallelism small in order to leave as much tolerance as possible for linear dimensions.

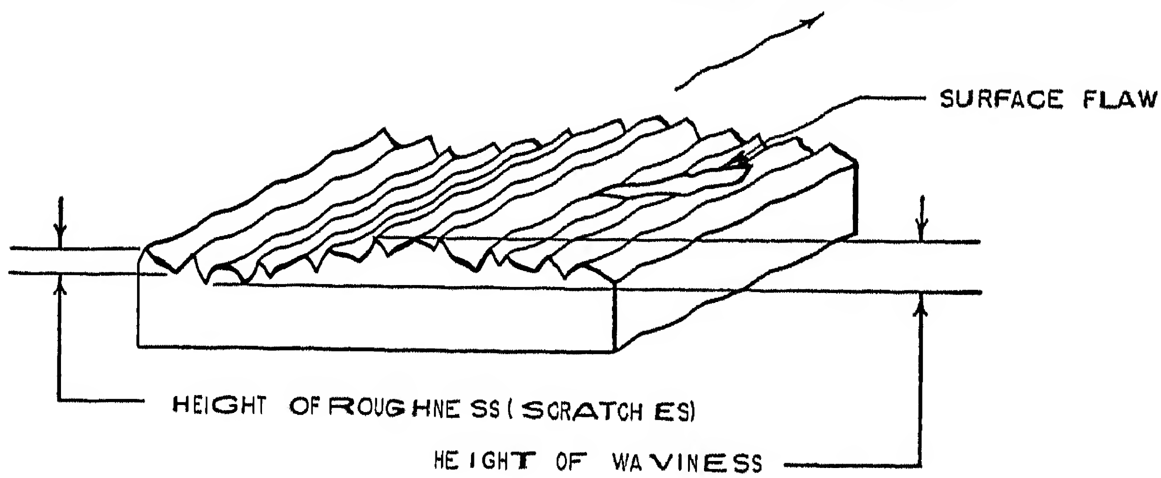


Fig. 2–4. A typical surface highly magnified.

Surface Quality

The nature of surface quality. Any machined surface is only an approximation of an ideal. Under sufficient magnification a machined surface is seen to be made up of a series of peaks or ridges and valleys. They are visible to the naked eye if the surface is quite rough. A typical surface is indicated by Fig. 2-4. Surfaces in general are very complex and result from several kinds of variations. The principal elements of surfaces have been defined by the American Standards Association in ASA B46.1-1947, as follows:

"*Surface.* The surface of an object is the boundary which separates that object from another substance. Its shape and extent are usually defined by a drawing or descriptive specifications.

"*Profile.* The contour of any specified section through a surface.

"*Roughness.* Relatively finely spaced surface irregularities. On surfaces produced by machining and abrasive operations, the irregularities produced by the cutting action of tool edges and abrasive grains and by the feed of the machine tool are rough-

ness. Roughness may be considered as being superposed on a wavy surface.

"*Waviness.* The surface irregularities which are of greater spacing than the roughness. On machined surfaces, such irregularities may result from machine or work deflections, vibrations, etc. Irregularities of similar geometry may occur because of warping, strains, or other causes.

"*Flaws.* Irregularities which occur at one place, or at relatively infrequent intervals in the surface, e.g., a scratch, ridge, hole, peak, crack, or check.

"*Lay.* The direction of the predominant surface pattern.

"*Microinch.* One millionth of an inch (0.000001 in.)"

Irregularities may vary in height, width, length, shape, and direction. The distance between the irregularities that give roughness to a smooth machined surface is ordinarily between 0.0002 and 0.010 in., with average height much less, usually between 0.00001 and 0.0005 in. The length of the undulations that cause waviness is generally in excess of 0.025 in. and may be as much as 1.0 in. These waves are frequently cyclical or regularly recurring.

The heights of the irregularities of roughness are measured in microinches. Figure 2-5 represents the profile of a surface highly magnified. A centerline is drawn through the profile representing

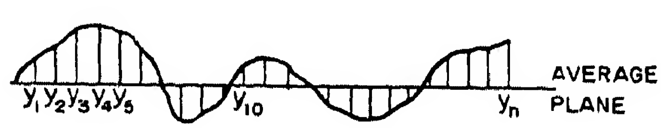


Fig. 2-5. A profile of surface roughness.

the average plane of the surface. The ordinates $y_1, y_2, y_3, \ldots . y_n$ show the variations of the profile from the centerline at equal intervals. A figure that is commonly used to depict the extent of the variations is the *root-mean-square* average of the deviations in microinches, abbreviated *rms.* To find the rms average of the profile of Fig. 2-5, the ordinates are squared and added, to give $y_1^2 + y_2^2 + y_3^2 + \ldots . + y_{n-1}^2 + y_n^2$. This sum is divided by the total number of ordinates. The square root of the quotient is the rms average of the profile. If the deviations are measured in microinches,

the rms average is expressed in the same units. This form of average is used because it gives more weight to the large deviations. Instruments that measure roughness trace a surface, measure an infinite number of ordinates, and compute the rms average automatically.

In addition to irregularities, other factors may enter into surface quality. Such factors include material, hardness, color, luster, and metallurgical structure. From the standpoint of metal machining, the irregularities and to some extent the color and luster are the important effects.

Surface finishes produced by metal machining. Each method of machining produces a generally characteristic surface. Operations like turning and shaping, in which tools cover the surfaces in continuous lines, leave a regular pattern of roughness. In grinding, a large number of cutting edges act on a surface, and although the pattern is often directional, the scratches vary in length and overlap to some extent. Operations like lapping and honing produce erratic crisscross patterns. Thus, the general appearance of a surface is usually determined by the method of finishing it.

Some machining operations are capable of producing very smooth surfaces. The roughness that may be expected from any operation lies within a range. The actual surface resulting from a specific application depends upon such factors as the condition of the machine and tools and the care and skill exercised by the operator. A comparison of ranges of surface roughness produced by common operations is made in Fig. 2-6.

Smooth surfaces are not always the best. A surface that does not bear against others does not generally need to be refined. A surface that is too smooth may not be a good bearing surface in some circumstances because it may be difficult to keep it lubricated. Some roughness to provide oil pockets is often desirable. On the other hand, the peaks and projections of too rough a surface wear away rapidly in service, and the surface is impaired. The smoother a surface must be, the more the cost to produce it. For economy, as rough a finish as will adequately meet functional needs is normally permitted.

Designations for surface finish. Surface quality requirements may be designated on drawings by conventional symbols described

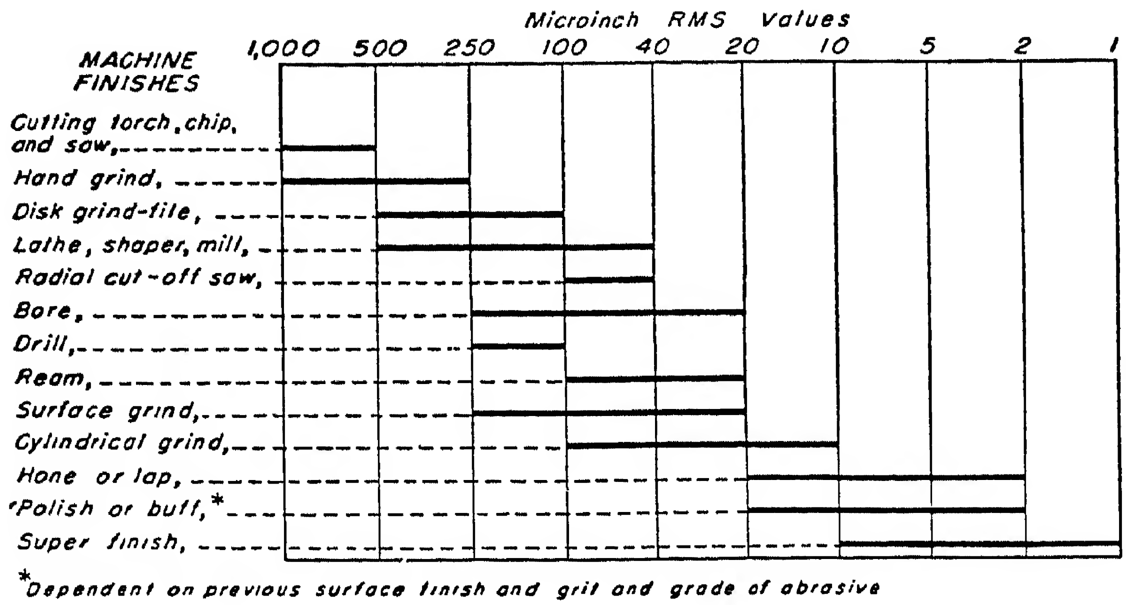
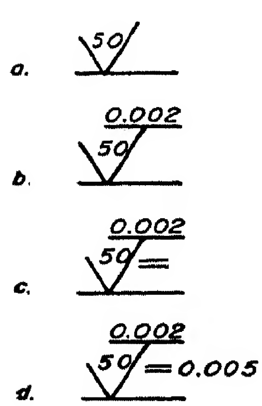


Fig. 2-6. Range of surface roughness produced by various machining operations. (From Spotts, Design of Machine Elements, **p. 335, Prentice-Hall, Inc., 1948.)**

below. In common practice, surface quality is specified only where a certain degree of finish is important.

SYMBOL EXPLANATION



Roughness height value (in microinches) is placed adjacent to and on the inside of the long leg of the vee directed upon the boundary line of the surface.

Waviness height value, when required (in inches), is placed above the horizontal extension line added to the long leg of the vee.

Lay designation, when required, is indicated by symbol placed under the extension.

Roughness width value, when required, is placed to the right of the lay symbol.

SYMBOLS INDICATING DIRECTION OF LAY

=	Parallel to boundary line of surface.
⊥	Perpendicular to boundary line of surface.
X	Angular in both directions to boundary line of surface.
M	Multi-directional.
C	Approximately circular relative to center of surface.
R	Approximately radial.

The plug of Fig. 2-7 is marked to illustrate the meanings of surface quality symbols. On the large diameter, the figure 1 inside of the $\surd$ specifies a roughness not to exceed 0.000001 in. rms (one microinch rms). The letter M designates that the scratches must not lie in any one direction, but must occur at random. The dimension of 0.0001 above the line specifies that the waviness height must not be more than 0.0001 in. from the troughs to the crests of the waves. The student should interpret the symbols on the small diameter of the plug.

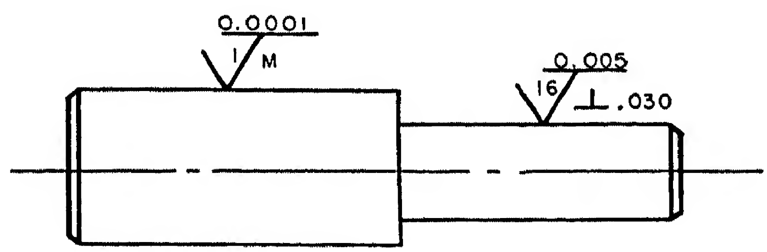


Fig. 2-7. A drawing of a plug showing surface quality specifications.

Questions

1. What is interchangeable manufacture? What benefits does it provide?
2. What are tolerance, allowance, basic size, standard size, nominal size, and limits?
3. What are a bilateral tolerance, a unilateral tolerance, and a limiting dimension?
4. Describe standard hole practice and standard shaft practice.
5. What determines how much tolerance should be assigned to a dimension?
6. What are standard fits? Of what value are they?
7. What are geometric dimensions?
8. Define roughness, waviness, flaws, and lay.
9. What effect do geometric tolerances have upon linear tolerances?
10. What is meant by rms?
11. How do the characteristics of surfaces depend upon the methods used to machine them?
12. How may surface finish be specified on a drawing?

Chapter 3

MEASURING INSTRUMENTS AND GAGES

ONE OF THE PRIME PURPOSES of metal machining is to produce objects having definite dimensions and shapes. To do that accurately, an operator must be able to observe and control dimensions closely. Measuring instruments and gages are the devices that enable a machinist to control his work and prove the results.

Uses and Types of Measuring Tools and Gages

As cuts are taken, dimensions must be measured to determine whether tools and machines are doing what is required. After workpieces are finished, they must be checked to ascertain whether they meet specifications. Measuring tools and gages are also helpful in other ways. They are used to lay out the positions of surfaces to be machined, to set and adjust tools, and to align machines.

A measuring tool is a device for determining the actual size of a dimension. Those that are capable of measuring within 0.001 in. are often called precision measuring instruments. Direct measurements are made by applying a reference directly to a workpiece. A scale placed between two points and observed gives a direct measurement. Direct measurement is generally more rapid but less accurate than measurement by comparison. Comparative measurements tell how much a dimension differs from a known dimension and are made by devices called comparators that magnify small distances so that they can be detected easily.

Gages are intended for quickly checking parts in production, to

avoid making actual measurements and to save time, and usually do not reveal the actual sizes of dimensions.

Some error must be accepted in all measuring and gaging. Wear, thermal expansion, defects in instruments, and human errors cause inaccuracies. Hard and wear resistant materials are placed at critical spots to minimize wear. All metals change in size with a rise or fall of temperature, and different metals expand and contract at different rates. For consistent results, a temperature of 68° F. has been established as the normal for making precise measurements. Measuring tools and gages are never perfect. The more precise a measurement must be, the more accurate must be the reference to which comparison is made. This leads ultimately to standard references that have such small inaccuracies that they are considered exact for practical purposes. Human judgment and skill enter to some extent when a measurement is made. The more precise a measurement must be, the more elaborate must be the equipment to reduce the effect of human error. All of the foregoing conditions account for the existence of a large variety of measuring instruments and gages.

Most measuring instruments have specific and limited uses, although some can be used for more than one purpose. Some are suitable for measuring linear dimensions, others for angular or geometric dimensions, and specialized ones are devoted to measuring surface finish. Certain tools are for marking surfaces to establish workpiece lines. Some measuring devices are reserved for reference purposes, as standards of comparison. A large variety of measuring tools is required to satisfy the many needs that arise in industry. Gages also are found in many forms and sizes. The principal measuring tools and gages and their uses will be described.

Linear Measuring Instruments

Steel rules. The standard steel rule or scale is a straight edge with graduations. Plain scales vary in length from less than one inch to four feet, are hardened and ground, and have etched or cut graduations. A rule commonly used by machinists is shown in Fig. 3-1. Several forms of rules are available and convenient for various purposes. These include narrow rules, thin flexible rules, and short rules with handles. Regular steel rules often have short

scales along their ends. Long flexible scales that may be rolled up for compactness are known as steel *tapes*.

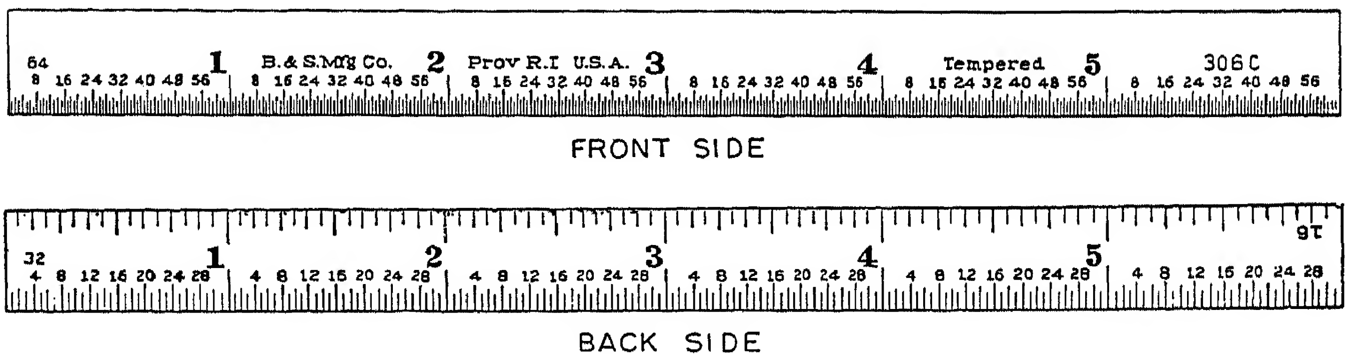


Fig. 3-1. A 6 in. steel rule. (Courtesy Brown and Sharpe Mfg. Co.)

Scales are graduated in both metric and English units. The latter is customary in the United States. Some scales are divided into tenths and hundredths of an inch, but the most common form of graduation is *binary*. That means that the inch is divided by 2, again by 2, and so on, giving the fractions of 1/2, 1/4, 1/8, 1/16, 1/32, and 1/64 in. Smaller divisions are not used because they are hard to read. A steel rule, like that in Fig. 3-1, generally has a scale along edges on both sides, each with different divisions.

Most people can read a scale only to the nearest 1/64th in., but experienced machinists can split a 64th. Normally, a steel rule is intended for measuring dimensions that do not have to be held more closely than the smallest graduation. The accuracy obtained from a scale depends upon the quality and condition of the scale and the skill of the user. Worn rules and indistinct graduations are not conducive to good results. To be kept in good condition, a steel rule should never be used for any purpose other than measuring.

Proper use of a steel rule involves several factors. If a shoulder or face fixes one end of a measurement, a scale may have to be abutted against the shoulder. In that case, the end of the scale should be held firmly in full contact with the surface. However, whenever possible a rule should be positioned so that a start can be made from a line on the scale, preferably the one-inch line, in reading a dimension. That eliminates the chance of error from the end of the rule being worn. Also, a line can be sighted more easily than the end

of the rule. As indicated in Fig. 3-2, a scale should be held against the surface and along the line where a reading is taken.

A *hook rule* has a right-angle projection on one end that can be brought up against a shoulder to position the scale. This is particularly convenient if a measurement must be made from an edge that is hidden. A hook on a rule also provides a convenient ledge to position inside calipers and dividers when they are set to a size from a scale.

A *depth rule* or *depth rule gage* is a narrow scale or rod with a sliding crosshead that can be clamped in any position. This scale is convenient for checking the depths of holes and slots. The rule may be set at an angle with respect to the head, as shown in Fig. 3-3.

Calipers. A caliper is used to transfer and compare a dimension from one object to another or from a part to a scale or micrometer where the measurement cannot be made directly. That also is the function of telescoping and small hole gages.

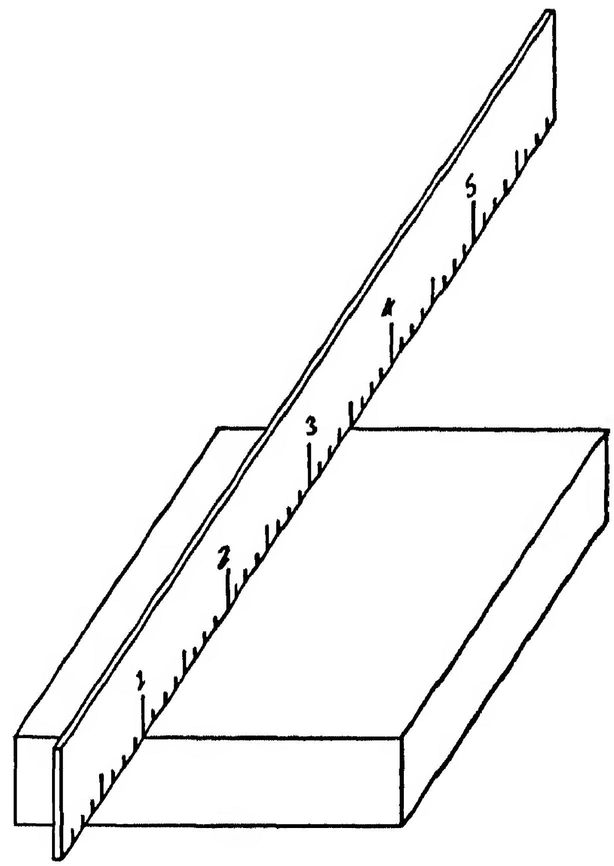


Fig. 3-2. Proper use of a steel rule.

Spring calipers are illustrated in Fig. 3-4. A loop spring on top of the joint between the two legs applies force tending to separate the legs at the bottom. An adjusting screw and nut keep the legs in position. The adjusting nut is split on many calipers so that it can be expanded and slid along the screw to an approximate position when the spring pressure is relieved by pressing the legs together. The nut is turned on the screw to make final adjustments.

An *outside spring caliper* has legs turned inward. It may be set to a size from a scale, a plug, or other reference and applied to a workpiece to see if the sizes agree. The caliper may be set to the work, and the dimension measured by a scale. When a caliper is applied to an object, it must make sure contact but not be forced. A sense of "feel" is necessary to use a caliper successfully. When an

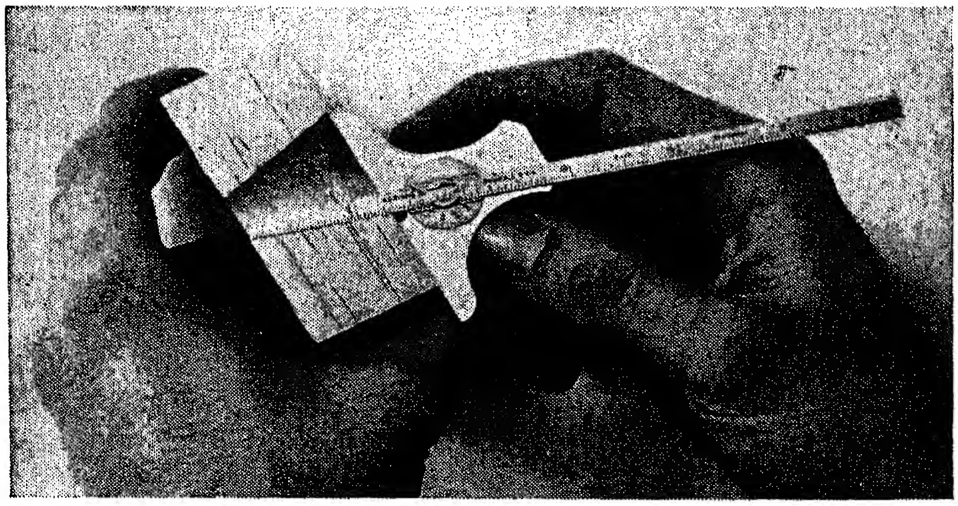


Fig. 3-3. Angle and depth checked with a depth rule. (Courtesy Brown and Sharpe Mfg. Co.)

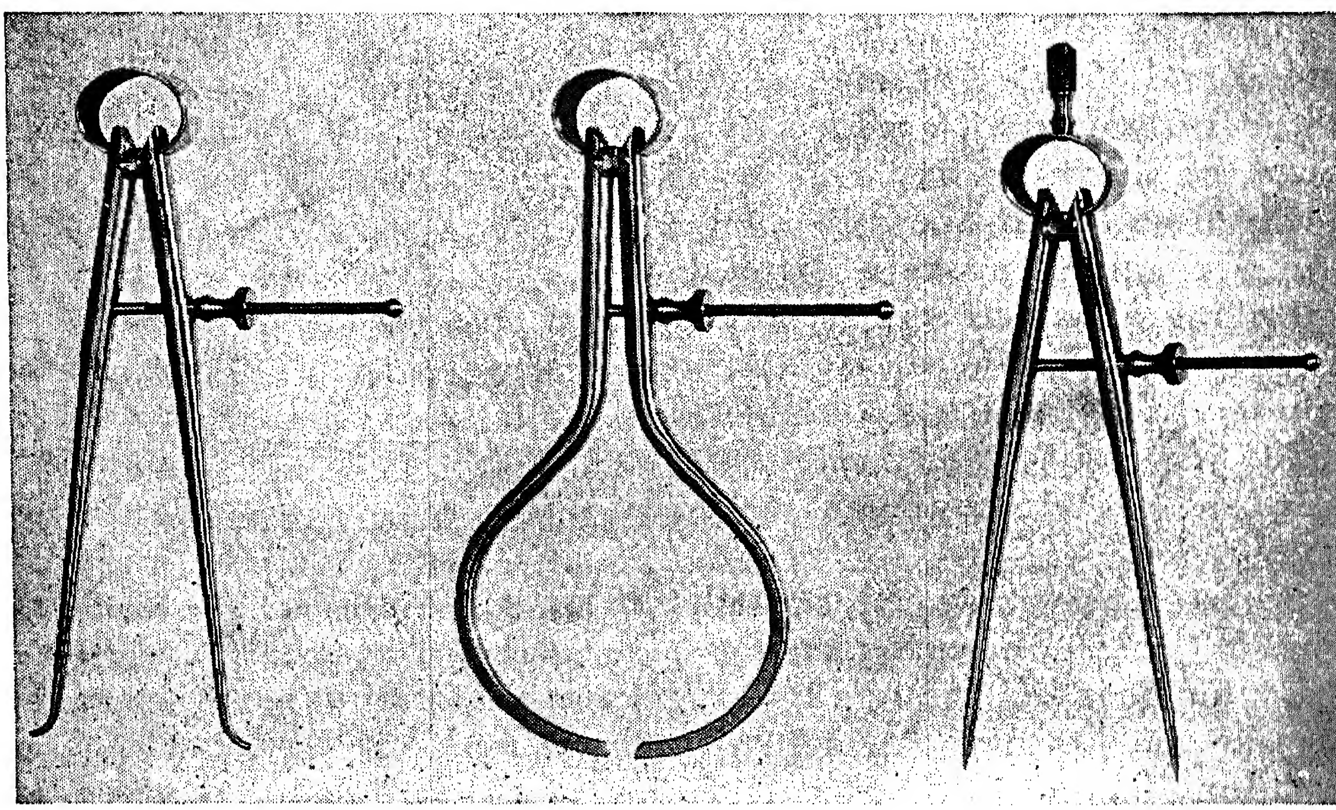


Fig. 3-4. Inside and outside calipers and dividers. (Courtesy The L. S. Starrett Co.)

outside caliper is set to a scale, one leg is held firmly against the end of the scale, and the other end is adjusted to the desired scale reading. A spring caliper is held in position by a contant spring tension and can be adjusted in fine increments, which permit full use of the sense of touch in measuring.

An *inside spring caliper* has the ends of its legs turned outward to make contact with the insides of holes and grooves. Inside calipers

are set and checked with ordinary scales having one end resting against a true surface. Hook scales, ring gages, and micrometer calipers may also be used to set this type of caliper.

Spring joint calipers are used for dimensions up to about 8 in., firm joint calipers for larger sizes. A *firm joint caliper* has a friction joint between its legs and no adjusting screw. Firm joint calipers are made in both inside and outside styles.

A *transfer caliper* is a kind of firm joint caliper with an arm shorter than and independent of the two legs. One of the legs can be clamped to the arm. When unfastened, the leg can be swung away without disturbing the arm. The transfer caliper is useful for checking inside recesses, as is done in Fig. 3-5. After a leg has been retracted and the caliper removed from the recess, the leg is moved back to the arm to restore the setting of the caliper.

A *hermaphrodite caliper* has a bent leg and a straight leg with a sharp point. It is actually a layout tool used to scribe a line parallel to an edge, as in Fig. 3-6, or to locate the center of a round bar.

Telescoping gages are shown in Fig. 3-7. The head of the T is hollow and contains a plunger that is pushed out by a spring. Some models have two plungers, one on each side of the head. The

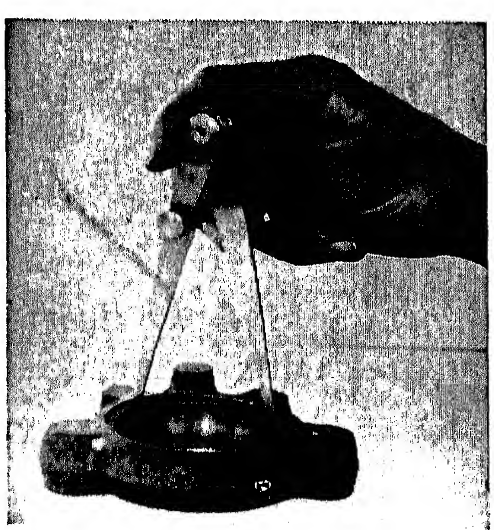


Fig. 3-5. A transfer caliper applied to an inside recess. (Courtesy Brown and Sharpe Mfg. Co.)

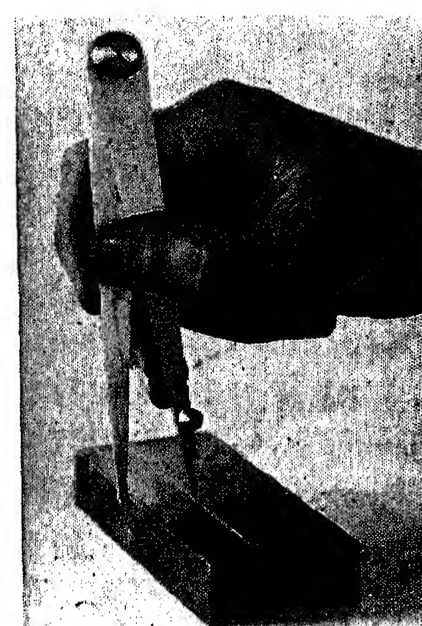


Fig. 3-6. A line being scribed parallel to the side of a block by a hermaphrodite caliper. (Courtesy Brown and Sharpe Mfg. Co.)

plunger or plungers can be locked in position by turning a knurled screw in the end of the handle. To measure the diameter of a bore, the T head is placed in the hole and allowed to expand to touch opposite sides. The gage is locked, taken out of the hole, and measured by a micrometer.

Telescoping gages come in sets with a range from 5/16 to 6 in. The handle is convenient for reaching into fairly deep holes.

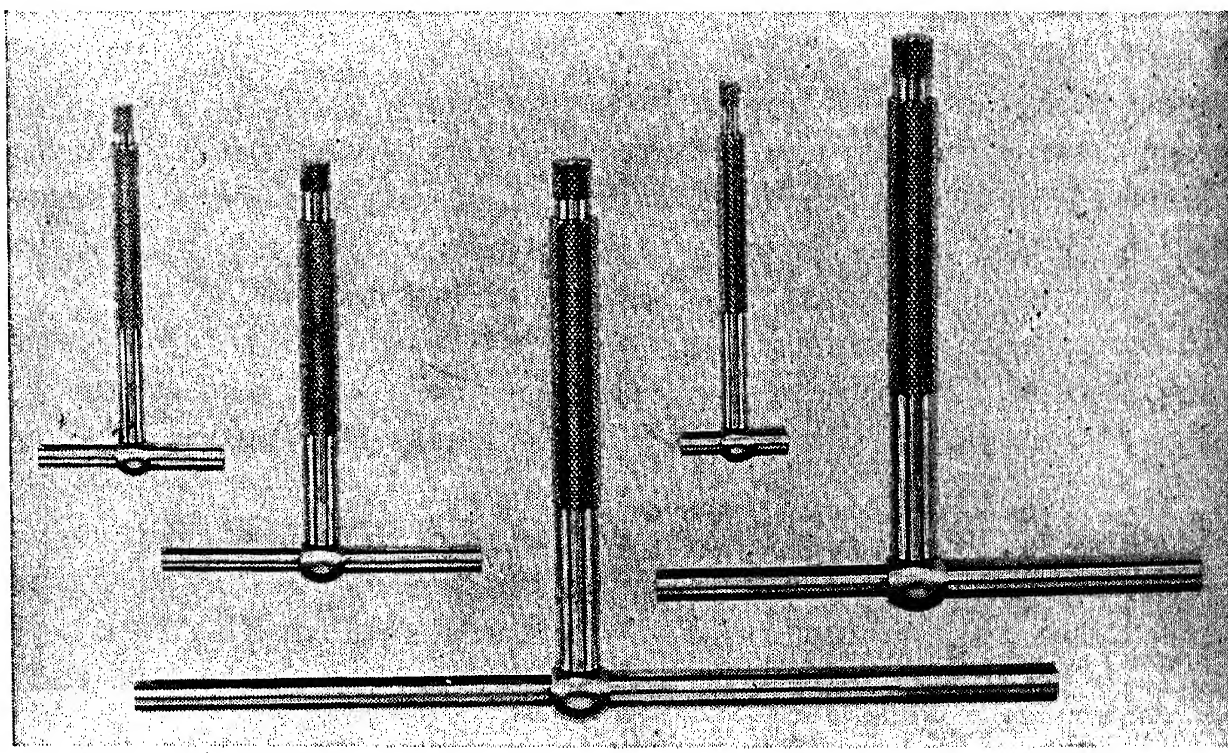


Fig. 3-7. Telescoping gages. (Courtesy The L. S. Starrett Co.)

A *small hole gage* is a version of the telescoping gage for holes, slots, grooves, and recesses from 0.125 to 0.500 in., too small or shallow for regular telescoping gages. On one end of the handle of a small hole gage is an expanding split knob or button, and on the other end is an adjusting screw.

Verniers. The vernier is based upon the principle of two differently graduated scales to measure more closely than with either alone. A

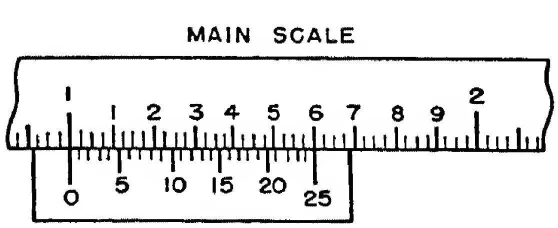


Fig. 3-8. A typical vernier scale.

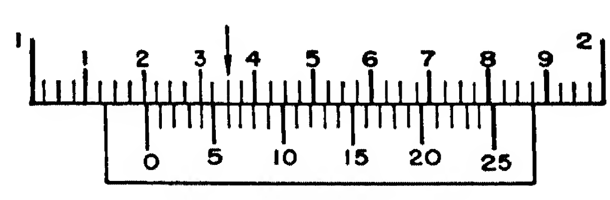


Fig. 3-9. A vernier scale reading 1.206 in.

typical vernier arrangement is shown in Fig. 3-8. One inch on the main scale is divided into ten equal parts, and each tenth inch into four parts. Each small space is thus 1/40 or 0.025 in. The vernier scale is 0.6 in. long and is divided into 25 equal parts. Each vernier scale space is thus 0.024 in., or 0.001 in. less than each space on the main scale. As shown, the No. 0 and No. 25 lines of the vernier scale are aligned with 1.0 in. and 1.6 in. lines respectively on the main scale. If the vernier scale is moved 0.001 in to right, its No. 1 line will coincide with the first line after 1 in. on the main scale, and no other lines will match. If the vernier scale is moved 0.002 in. to the right, the next two lines will coincide, and so on.

The position of a vernier scale is always designated by its O mark. The number of thousandths that it is beyond the line just passed on the main scale is signified by the line on the vernier scale that

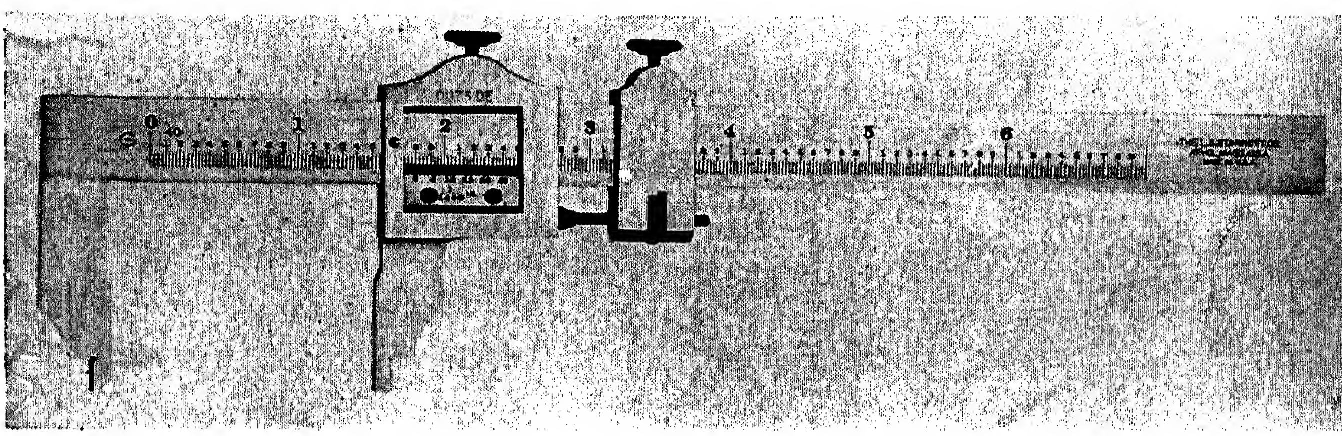


Fig. 3-10. A vernier caliper. (Courtesy The L. S. Starrett Co.)

matches a line on the main scale. As an example, in Fig. 3-9 the O mark of the vernier scale has just passed 1.200 in. on the main scale. Also the No. 6 line of the vernier scale coincides with a line of the main scale. Thus the dimension reads 1.206 in.

The *vernier caliper* consists of a graduated steel rule with a fixed jaw on one end and a sliding jaw with a vernier scale. The vernier caliper of Fig. 3-10 can be used for outside or inside measurements. Outside dimensions are measured between the jaws, inside dimensions over the tips of the jaws. On one side of the sliding jaw is a vernier scale for outside dimensions, on the other side a vernier scale for inside dimensions. The scales are graduated like that of Fig. 3-8 and can be read to 0.001 in. The sliding jaw has two sections connected by an adjusting screw. Either or both sections can be locked in place. For a measurement, the moveable jaw is slid to

an approximate position, the outer locking screw is tightened, and the final adjustment is made with the adjusting screw. When the proper setting has been reached the other locking screw is tightened to keep the vernier scale from moving. Center points, one on each jaw, are provided for setting dividers. Dependable measuring with a vernier caliper requires an instrument in good condition properly used.

A plain *slide caliper* resembles a vernier caliper but has no vernier

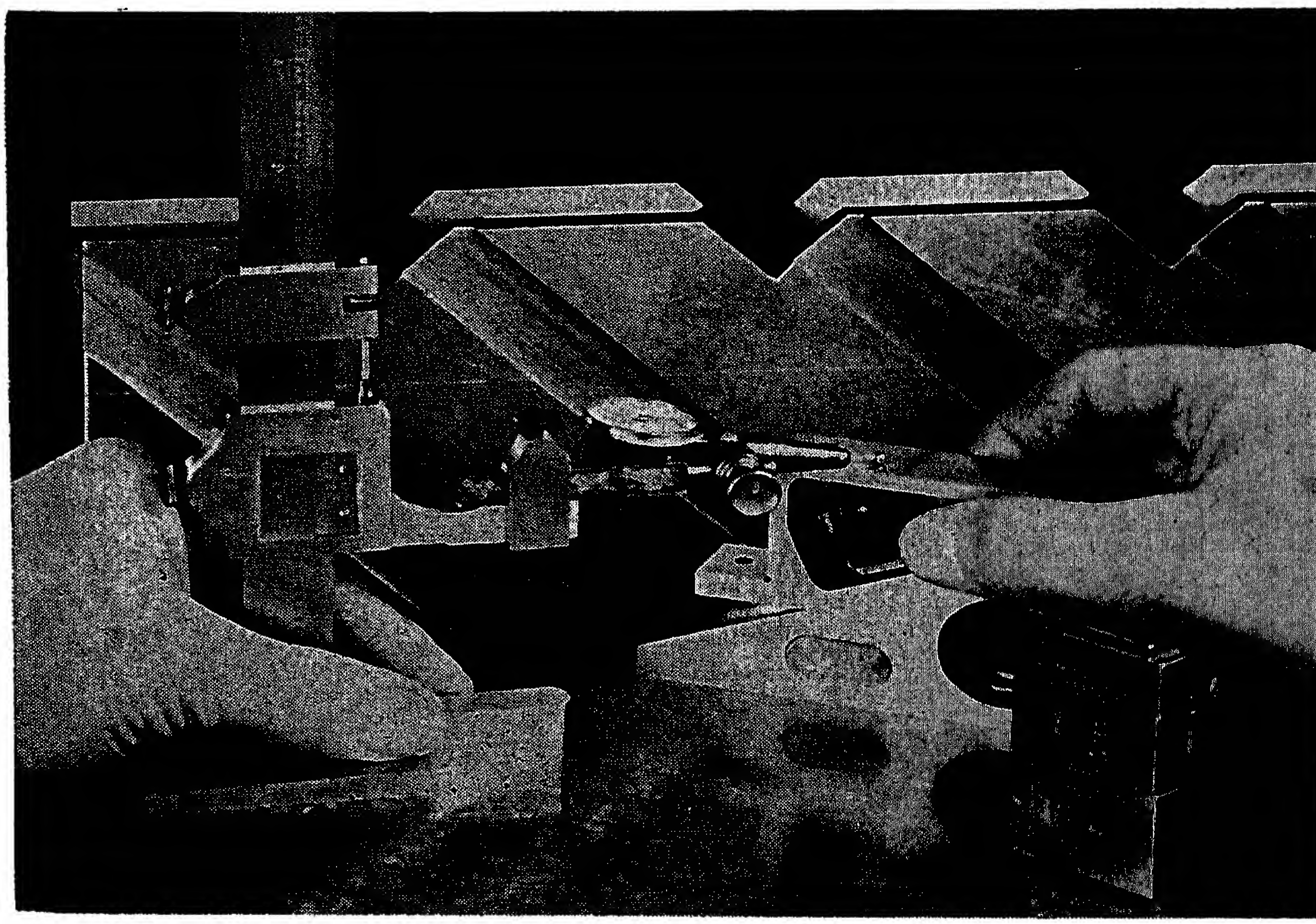


Fig. 3-11. A vernier height gage and a dial indicator being used to set a planer gage from a stack of gage blocks. (Courtesy The L. S. Starrett Co.)

scale. Readings for both inside and outside measurements are made directly from a single scale.

The *vernier height gage* is like a vernier caliper. The fixed jaw is the base as shown in Fig. 3-11. The sliding jaw may have a scriber clamped to it for layout work. A dial indicator or depth gage attachment may be attached to the movable jaw for making measurements or comparisons, as is done in Fig. 3-11.

A *vernier depth gage* is like a depth rule gage illustrated in Fig. 3-3 in that it is made up of a rule and a sliding crosshead. In

addition, a vernier depth gage has a vernier scale on the crosshead and an attachment for fine adjustments like the vernier caliper. This instrument can be used to measure the depths of holes and slots to 0.001 in.

Micrometers. A *micrometer caliper* is a sliding caliper that is adjusted accurately by a precision screw and has a scale that can be read to 0.001 in. or less. A cross section of a typical micrometer is shown in Fig. 3-12. Measurements are made between the fixed anvil and the movable spindle carried by the frame. The spindle is an unthreaded extension of the screw and is moved by turning the

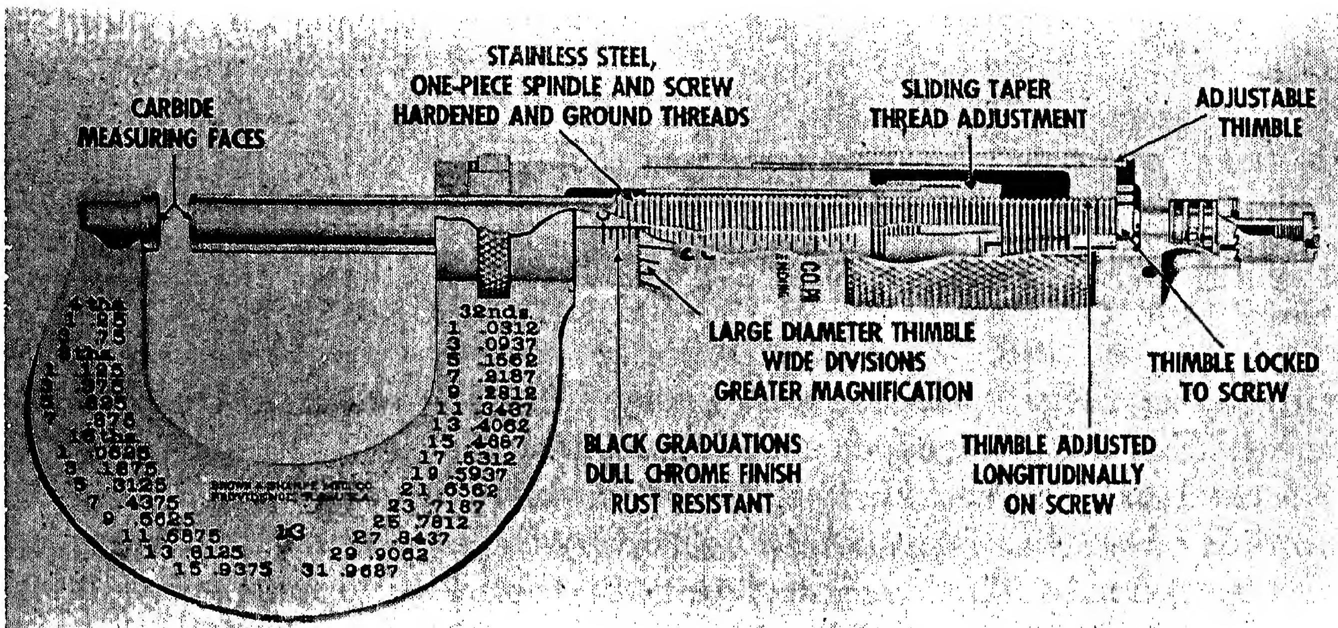


Fig. 3-12. A sectional view of a micrometer caliper. (Courtesy Brown and Sharpe Mfg. Co.)

thimble or sleeve. The nut is held in the barrel on the frame. The barrel and thimble carry the micrometer scales and overlap to keep dirt out of the mechanism.

The small knurled extension on the end of the thimble is convenient for turning the spindle rapidly in changing its position. This extension on some micrometers is also connected to a ratchet stop for applying a uniform pressure in measuring. A clamp ring in the frame around the spindle can be turned to lock the spindle in any position.

The customary pitch of the screw of a micrometer is 1/40 or 0.025 in. That means the screw and spindle move lengthwise 0.025 in. for each complete revolution. The lines on the barrel are spaced apart an amount equal to the pitch of the screw. Thus, each line on

the barrel indicates 0.025 in. Each fourth line is longer, and the long lines are numbered 0, 1, 2, 3, 4, 5, 6, 7, 8, 9, 0. Four divisions of 0.025 each are equal to 0.100 in., and the numbers represent 0.100, 0.200, 0.300, 0.400, 0.500, 0.600, 0.700, 0.800, 0.900, and 1.000 in. The number of revolutions by which the spindle is separated from the anvil is shown by the number of divisions on the barrel uncovered by the thimble. The beveled edge of the thimble is marked

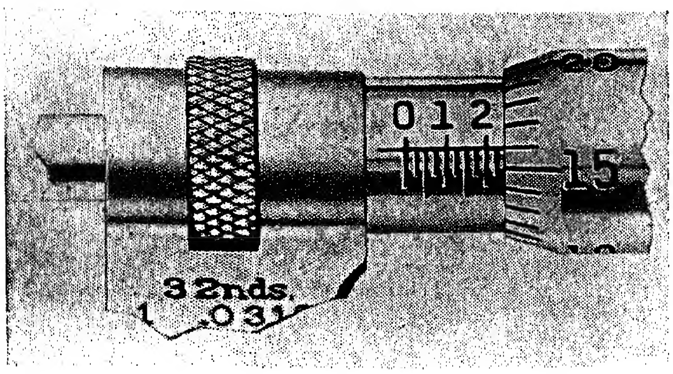


Fig. 3-13. A micrometer reading of 0.241 in. (Courtesy Brown and Sharpe Mfg. Co.)

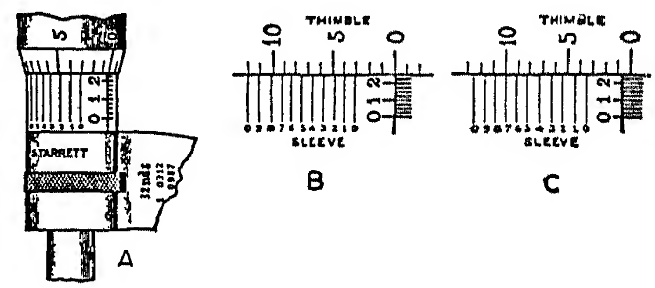


Fig. 3-14. The scales on a vernier micrometer caliper. (Courtesy The L. S. Starrett Co.)

with 25 equally spaced lines, every fifth one extended and numbered. Each of the spaces stands for 0.001 in.

A micrometer reading is the sum of three figures. The first is the number of tenths of an inch indicated by the largest exposed numeral on the barrel. The second figure is found by multiplying 0.025 by the number of small lines exposed on the barrel after the last numeral. The third figure in thousandths of an inch is equivalent to the number of the line on the thimble aligned with the lengthwise line on the barrel. These directions are followed in reading the micrometer setting of Fig. 3-13 in the following manner:

The largest exposed numeral on the barrel is 2 and represents	0.200 in.
After the numeral 2, one exposed line times 0.025 equals	0.025 in.
The sixteenth line on the thimble is aligned with the lengthwise line on the barrel and stands for	0.016 in.
The correct micrometer reading is	0.241 in.

Vernier micrometer calipers have a vernier scale on the barrel so that readings can be made to 0.0001 in. Typical readings are shown

in Fig. 3-14. Ten divisions around the barrel occupy the same space as nine divisions on the thimble. Thus the difference between one space on the barrel and one on the spindle stands for 0.0001 in. A vernier micrometer is first read to 0.001 in. like a regular micrometer, and then the vernier scale is read. In Fig. 3-14 B, the zero on the thimble is aligned with its reference line on the barrel, and the lines of the vernier scale marked *O* coincide with lines on the thimble. The reading is therefore an even 0.2500 in. In Fig. 3-14 C, the zero line on the thimble has gone beyond the reference line on the barrel. The seventh line of the vernier coincides with a line on on the thimble. Thus, the second reading is 0.2507 in.

The micrometer caliper, often called a "mike," is used to measure outside dimensions. It is simple, fast, and reliable if handled properly. The commonest size is the one-inch micrometer that serves for all dimensions from zero to one inch. Other sizes are available for dimensions up to 24 in., but each size has a normal range of only one inch. The size designation of a micrometer indicates the largest size it is intended to measure. For example, a 4 in. micrometer measures dimensions from 3 to 4 in. Some micrometers have interchangeable anvils to enable them to cover wide ranges.

Micrometer calipers are made in various forms to suit specific purposes. A micrometer with a deep throat is used for checking sheet metal. Some micrometers are mounted on rigid bases for bench use. For most work, micrometers have flat surfaces on the ends of their anvils and spindles. Wide-faced anvils and spindles are available for soft and resilient materials like paper and cloth. Some are made with balls, curves, edges, or points on the ends of anvils and spindles for checking irregular surfaces like grooves or threads.

A micrometer caliper must be kept in good condition if it is to give reliable measurements. The ends of the anvil and spindle must be parallel and just meet when the zero on the thimble coincides with the zero on the barrel. A micrometer should be checked from time to time at various positions with accurate gage blocks or rolls and adjusted if not right.

A micrometer caliper must be used carefully if accurate results are to be obtained from it. A small loose workpiece is held in one hand, the micrometer is held against the palm of the other hand by one or more fingers, and the thimble is turned by the thumb

and forefinger as is done in Fig. 7-4. For large or stationary work, the micrometer frame is held by one hand near the anvil, and the thimble is supported and turned by the other hand. In all cases, the micrometer must be held square with the work surfaces and slid slowly back and forth to get the correct "feel." The anvil and spindle must make positive contact with the work surfaces, but care must be taken that no force is applied that might spring the micrometer. These same precautions must be observed in using all measuring instruments, particularly calipers and gages. Tests have indicated that many mechanics do not manipulate micrometers carefully enough to justify readings to ten thousandths of an inch.

A micrometer caliper with a built-in dial indicator, called a *dial micrometer caliper,* is shown in Fig. 3-15. Pressure on the anvil moves the needle across the scale in the frame. The anvil can be retracted by pushing a button on the frame. At a uniform pressure, the needle returns to the same place on the scale and size can be read from the micrometer scale. This instrument helps eliminate the errors sometimes

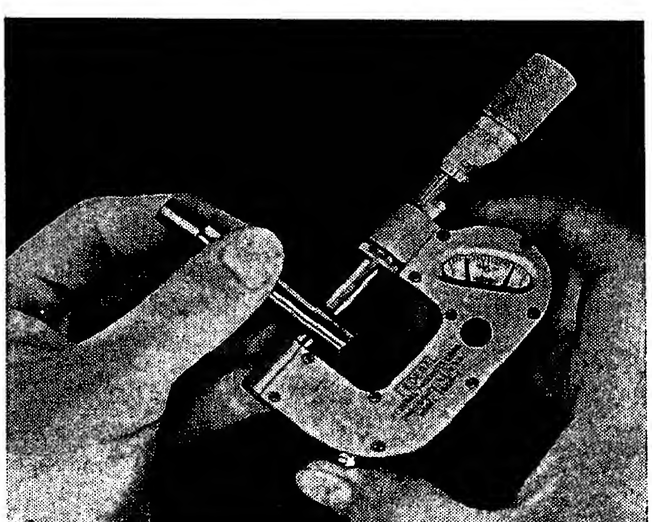


Fig. 3-15. An indicating micrometer. (Courtesy Federal Products Corp.)

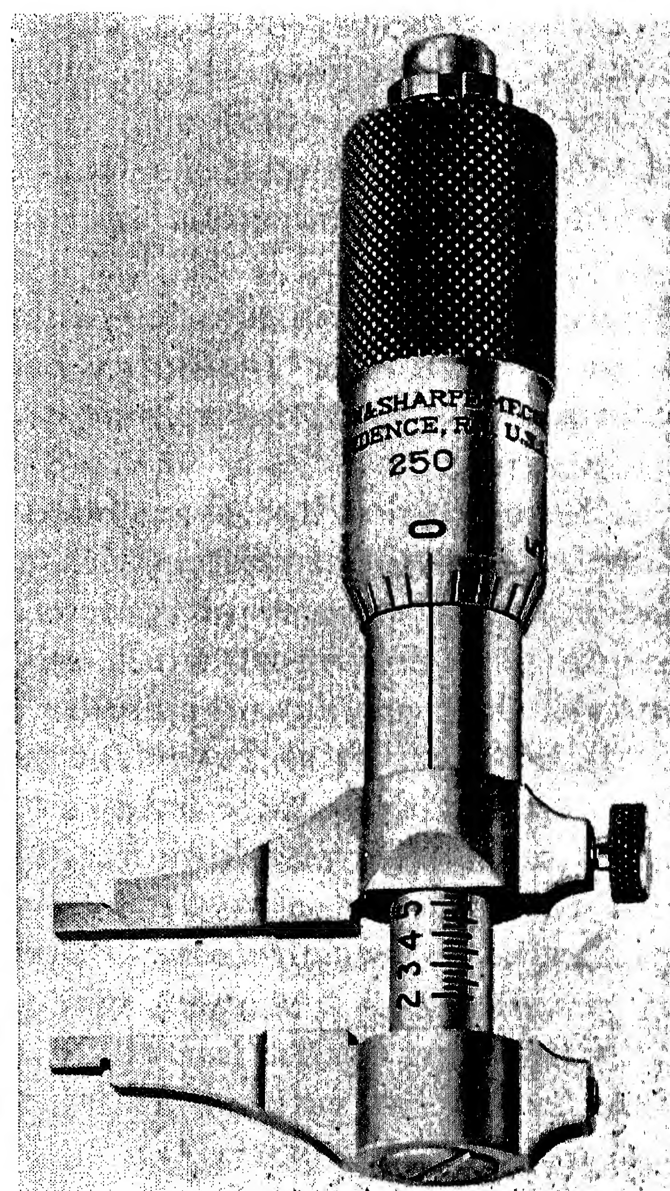


Fig. 3-16. An inside micrometer caliper. (Courtesy Brown and Sharpe Mfg. Co.)

caused by applying improper pressure to ordinary micrometer caliper. The spindle can be locked in place and the instrument applied as an indicating snap gage.

The *inside micrometer caliper* of Fig. 3-16 operates on the same principle as the outside micrometer caliper. It is used to measure diameters of holes and grooves and has the advantage of giving readings directly, unlike telescoping or small hole gages which must be checked by a micrometer. The use of inside micrometer calipers is limited to dimensions up to about two inches, where it is not convenient to try to insert an inside micrometer.

A typical *inside micrometer* consists of a holder with a micrometer screw, a spacing collar, and six extension rods that can be attached to the holder to measure from two to eight inches by thousandths of

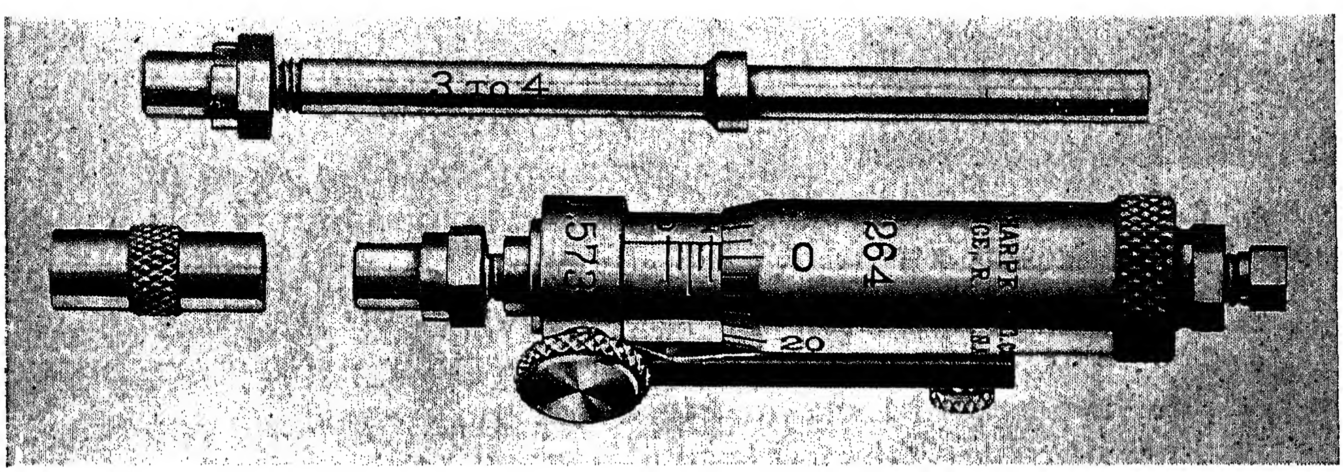


Fig. 3-17. An inside micrometer. (Courtesy Brown and Sharpe Mfg. Co.)

an inch. A holder, spacing collar, and extension rod for 3 to 4 in. dimensions are shown in Fig. 3-17. The extension rod for 2 to 3 in. dimensions is in place in the holder and gives a range of 2 to 2½ in. by itself. If the spacing collar is slipped over the end of the rod, the range is 2½ to 3 in. Each extension rod is provided with an adjustment to compensate for wear. Other models of inside micrometers cover various ranges. With a suitable base, the inside micrometer can be used for a height gage.

The reading of an inside micrometer can be conveniently checked with a micrometer caliper. This is usually done to assure correct readings whenever extension rods are changed in the head.

Holes from ½ to 8 in. diameter may also be measured with *internal micrometer plugs,* sometimes called micrometer plug gages. Seg-

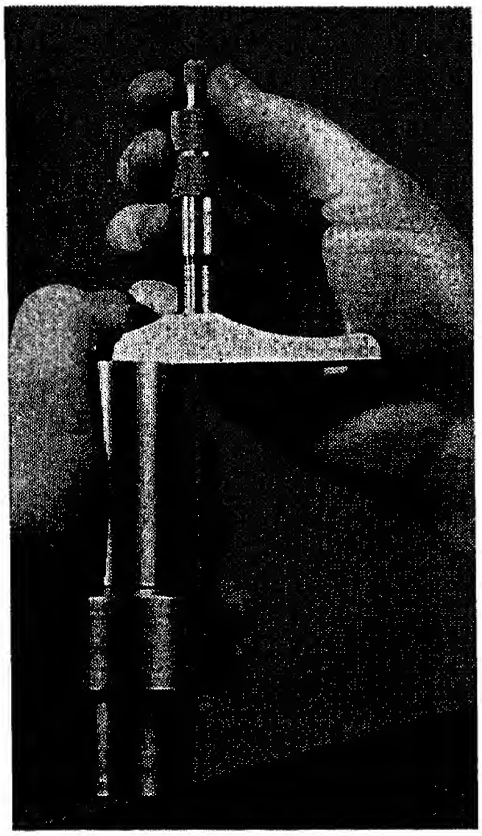


Fig. 3-18. An application of a micrometer depth gage. (Courtesy Brown and Sharpe Mfg. Co.)

ments of the plug are expanded by a taper, and the size is read on a micrometer scale.

A *micrometer depth gage,* like that of Fig. 3-18, has a micrometer barrel mounted on a base. The micrometer screw may have a travel of either ½ or 1 in. The spindle is hollow and takes extension rods that extend beyond the shoe for different ranges of measurement. With a set of rods, a typical over-all range for a micrometer depth gage is 0 to 9 in.

Dial indicators. Dial indicators or *dial gages* are devices that magnify small dimensional variations by means of gears and pinions or levers and show the variations on an easily read graduated dial. They are members of a class of instruments known as comparators that will be described later. A dial indicator can make no measurements by itself but can do so in connection with other instruments, like the gage blocks in Fig. 3-11.

Dial indicators are adaptable to almost all kinds of measuring and gaging. They are commonly applied to checking geometric accuracy. A typical application is that of Fig. 3-19, where a dial indicator with an attachment is set up for checking the runout of the bore of a piece mounted in a chuck. Dial indicators are used for checking machines and tools as well as workpieces. They aid in checking the alignment of centers, the runout of arbors, mandrels, and cutters, and the accuracy of movements of machine tool tables and rams.

Dial indicators are added to other measuring instruments and gages to make them easier to operate with less skill. An example is given by the indicating micrometer of Fig. 3-15. Other examples are indicating snap gages, calipers, bore gages, thickness gages, and depth gages. In addition, dial indicators are often incorporated into special gages and gaging fixtures for production inspection.

Many gages and measuring tools depend upon the sense of touch of the operator and require skill. Dial gages are superior in this respect because they magnify differences and show them in a way

that can be seen and evaluated. The initial cost of dial indicators is higher than that of many other tools but lower than elaborate electrical, air, and other comparators. Dial gages can be moved about easily and used under many conditions and in many places. In the long run they often cost less than cheaper instruments.

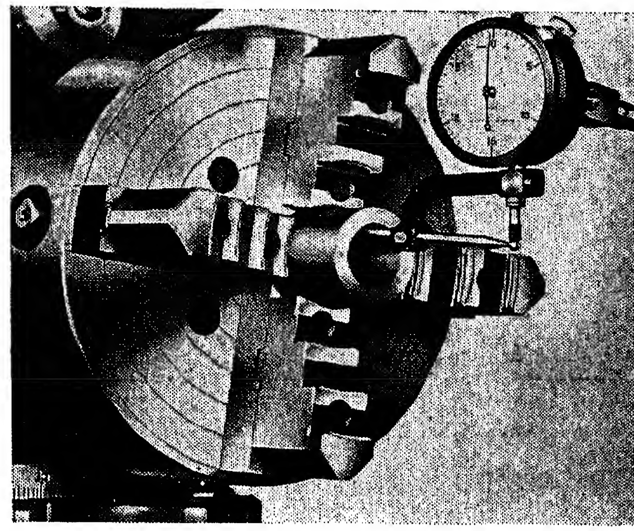


Fig. 3-19. A dial indicator with an attachment for checking the runout of the bore of a piece held in a chuck. (Courtesy Brown and Sharpe Mfg. Co.)

A dial indicator looks like a watch. A stem extends from one side and has a contact point on its end. Movement of the stem is transmitted and increased through levers and gears to turn the hand on the face of a dial. The mechanism is subject to the forces of springs that maintain a light but uniform pressure at the point of contact and eliminate backlash. Indicator mechanisms must be accurate and sensitive to react immediately to minute changes of size. Only a very small force is needed to activate an indicator. That is desirable because a large force would give rise to distortion and misleading readings.

Indicators are made in many styles and models. Dial scales are graduated in 0.0001, 0.00025, 0.0005, and 0.001 in. units, depending upon the ranges and magnifications of different indicators. The back of an indicator usually contains a lug or some other means for mounting the indicator. A variety of backs is available for various applications. Many other modifications and attachments can be supplied to accommodate indicators to a large variety of jobs.

An indicator may be mounted on a surface gage, on the movable jaw of a height gage, or on an element of a machine or tool such as a tool post or arbor. Several kinds of stands are made especially to hold indicators. In addition, many special kinds of supports are arranged for indicators, particularly on gages.

Angle Measuring Instruments

Combination square. A combination square consists of a center head, protractor, and square mounted together or separately on a

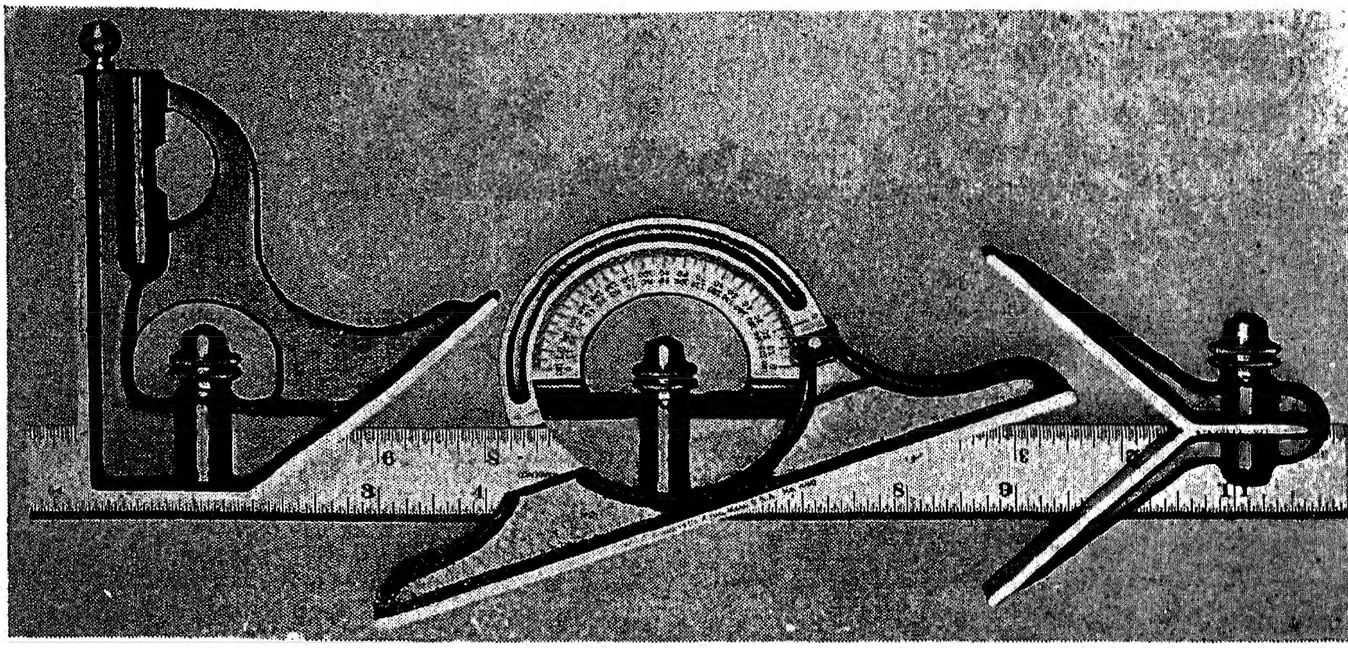


Fig. 3-20. A combination set. (Courtesy The L. S. Starrett Co.)

steel rule as shown in Fig. 3-20. A lock screw on each head is engaged in a slot in the rule and clamps the head against the straight edge at any desired position along the rule. The heads are normally mounted separately. The center head and rule provide a means for scribing lines to locate the center of a round piece quickly. The square head acts as a base to set the scale at 45° or 90° with a surface. The scale can also be set for depth measurements from the

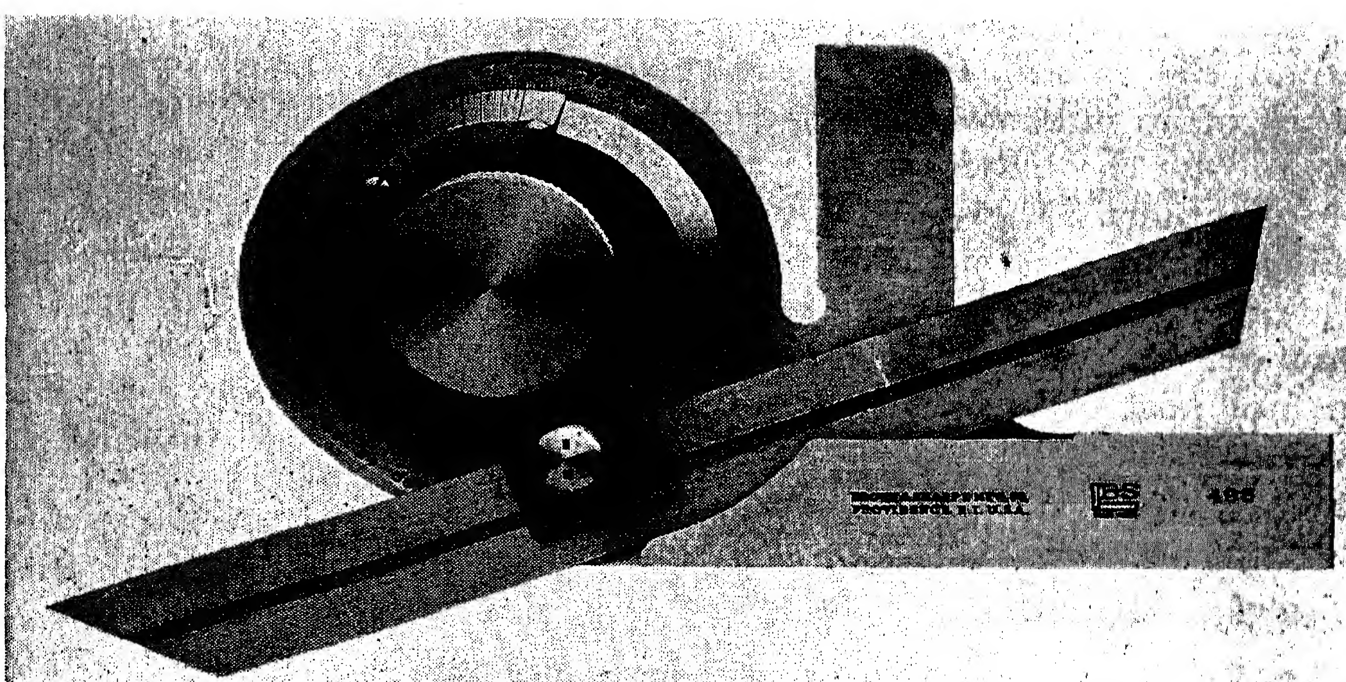


Fig. 3-21. A bevel protractor. (Courtesy Brown and Sharpe Mfg. Co.)

square. The protractor straight edge can be adjusted and clamped at an angle with the steel rule. This instrument is used for layout as well as measuring. The square head contains a small scriber and level.

Bevel protractor. A bevel protractor shown in Fig. 3-21 has a vernier scale for reading angles to five minutes of angular arc. The blade takes the angle set on the dial and can be moved back and forth over its entire length and clamped.

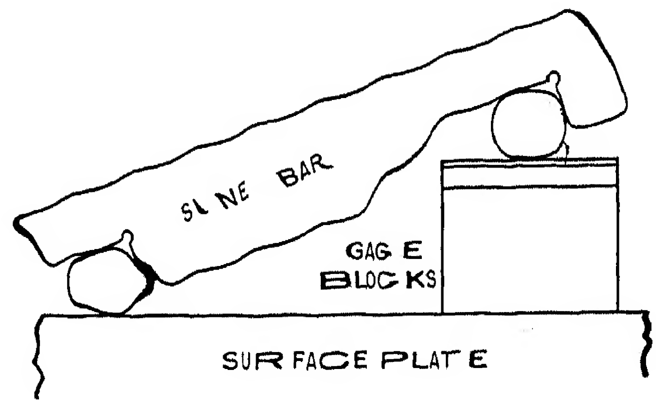


Fig. 3-22. Set up of a sine bar.

Sine bar. A sine bar provides the means of checking angles very closely. It consists of a bar with two rolls attached, as indicated in Fig. 3-22. The rolls are the same diameter, and their centers are spaced a definite distance, usually 5 or 10 in., and are on a line parallel to the top surface of the bar. One of the rolls is placed on gage blocks equivalent in height to the sine of the angle at which the bar is inclined.

Layout Instruments

Layout or *laying out* means to mark straight lines, circles, centers, etc. upon the surfaces of an object to serve as guides in finishing the piece. The accuracy of the lines determines the accuracy of the finished surfaces, and therefore the lines must be as exact as possible. Layout requires considerable skill and is normally done only when a comparatively few pieces of one kind are to be made.

Surfaces on which lines are to be marked are treated to make the lines stand out clearly. Rough surfaces may receive a coating of chalk or white lead. For fine layouts, smooth surfaces are copper coated or dyed with Prussian blue. Lines are literally scratched into the surfaces with sharp tools guided by other instruments. Some of the instruments already described are often used in layout work.

Marking tools. Hand marking tools include the sharp pointed *scriber* or *scratch awl*, the *prick punch* for light indentations, and the *center* punch for deeper marks.

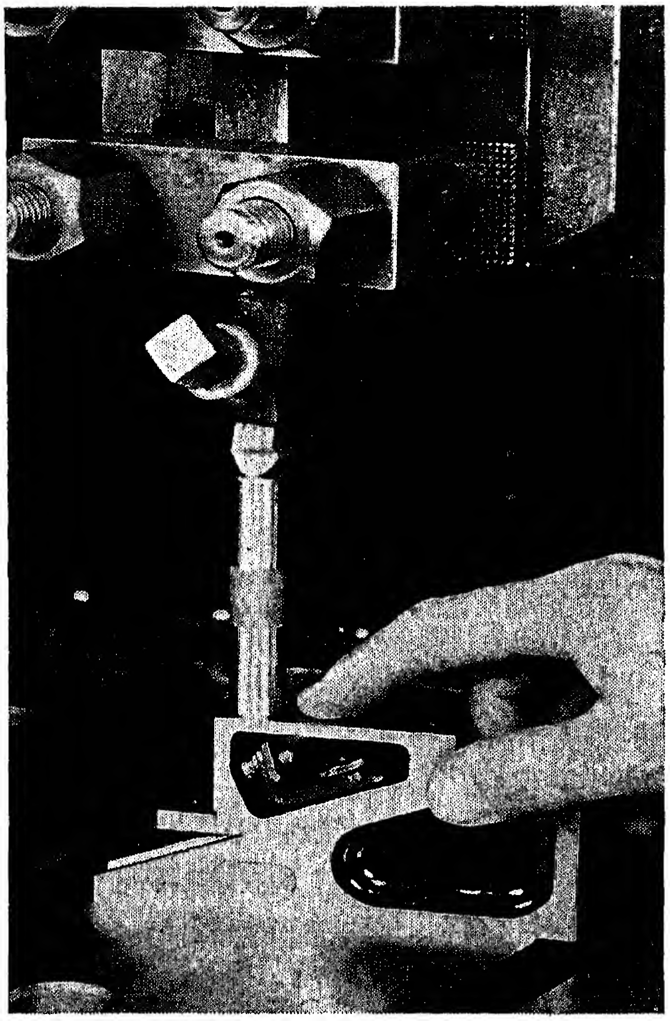


Figure 3-23. A planer tool being set by means of a planer and shaper gage. (Courtesy The L. S. Starrett Co.)

Spring dividers are shown in Fig. 3-4 and have straight legs ending in sharp points. They are used to transfer and compare distances and to scribe circles and arcs. For distances over about 10 in. a *trammel* is preferred. It has two legs with sharp points mounted and adjustable on a beam.

Planer and shaper gage. A planer and shaper gage consists of a tapered base and slide and an extension. Such a gage is illustrated in Fig. 3-23 in one of its principal roles, that of setting the tool on a planer. The gage also is used in layout work to establish heights and other distances. The distance between a face of the slide and a parallel side of the base can be varied by moving the slide along the taper. This distance has a range of ¼ to 8¼ in. Reference can be made to either side of the base. Measurements can be made from the extension bar on which the tool is resting in Fig. 3-23 or from the surfaces of the slide alone. The gage is adjusted to a micrometer, surface gage, height gage, caliper, or gage blocks as shown in Fig. 3-11.

Surface gage. A surface gage consists of an upright arm attached to a heavy base and an adjustable scriber on the arm. The arm of the universal surface gage of Fig. 3-24 can be inclined and adjusted to any angle. The base is grooved so it can be located on round pieces. The arm may be removed and the scriber clamped directly to the base.

Surface plates and accessories. True reference planes are necessary for accurate measuring and layout. These are furnished

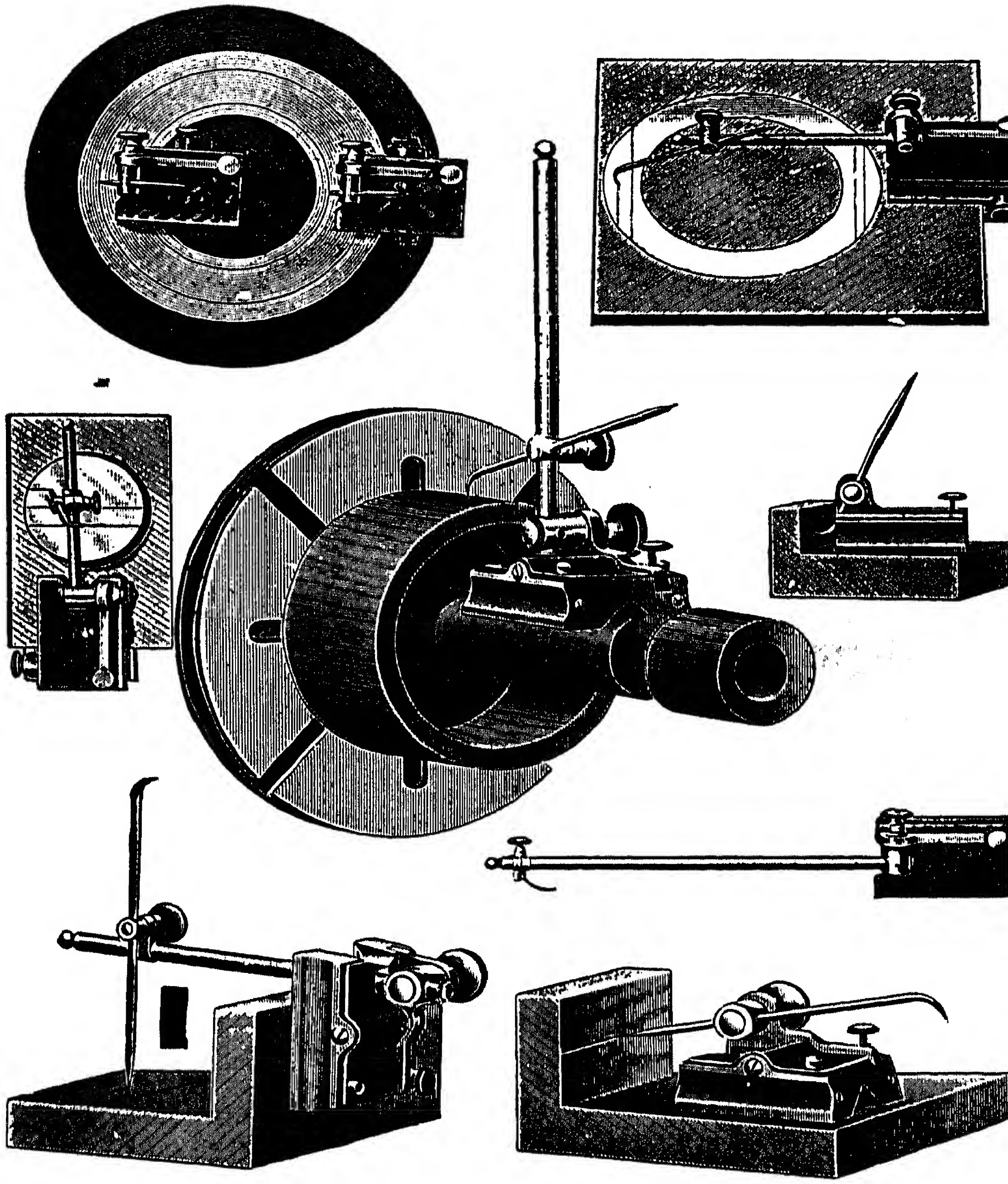


Fig. 3-24. A few applications of surface gages. (Courtesy The L. S. Starrett Co.)

by surface plates, angle irons, parallels, straight edges, and other accessories.

A surface plate is a heavy ribbed boxlike casting that stands on three points and has a thick and well-supported flat top plate. Its top surface is composed of a multitude of bearing spots all essentially in one plane. New plates generally have an average of eighteen bearing spots per square inch that do not vary from a true plane by more than 0.0002 in. per foot.

Cast iron surface plates are made in sets of three. They are first machined and then compared and corrected. When two metallic surfaces lightly coated with red lead or Prussian blue are rubbed

together, the areas where they bear upon each other are revealed. These are the high spots and are well distributed if the surfaces match. A few spots alone show that the surfaces do not coincide. The high spots are reduced by hand scraping away small particles of metal. The plates are compared repeatedly and corrected until they match. With only two plates, an unevenness of one surface can very well be complementary to another, but three plates do not match unless they are all true planes.

Most surface plates are cast iron, but some are granite. They range in size from 8 by 10 in. to 48 by 144 in.

Workpieces may be placed directly on a surface plate, supported on parallels, or clamped to the vertical surfaces of angle irons. Tools like height and surface gages can be moved about on a surface plate and still be kept in a definite relationship to a workpiece. In Fig. 3-11 a surface plate is used for accurate work.

A *toolmaker's flat* is a hardened steel disk about 1 to 6 in. in diameter and ¾ in. thick that has two sides flat and parallel within 0.00001 in. It furnishes very accurate reference surfaces for small, highly precise work.

An *angle iron* is an L-shaped casting having two or more true surfaces at right angles to each other. Angle irons come in many different sizes and proportions for various types and sizes of work. Workpieces can be clamped in vertical positions to angle irons and shifted from one position to another on a surface plate.

Bar parallels are bars of hardened alloy steel with sides ground to definite dimensions and held square and parallel, commonly within 0.0001 in. in six inches. They vary in length from 6 to 12 in. and in width from 1⅛ to 3 in. To insure accuracy, bar parallels are made and used in pairs. Larger parallels are called *box parallels*. They are made of cast iron and are hollow to reduce weight. Parallels are used to support and level workpieces on surface plates when the workpiece surfaces are irregular or uneven. They may also serve to raise a height gage or other instrument so that it will be effective above its normal range.

V blocks are special forms of parallels that have accurate V-shaped grooves for locating round pieces. They are made in various sizes and grades of accuracy and matched in pairs.

Reference Instruments

Precision gage blocks. Gage blocks are standards of accuracy, pieces of hardened steel or cemented carbide that represent definite

dimensions. The top and bottom surfaces of a gage block are not only a specified distance apart but also are lapped true, flat, and parallel within millionths of an inch. Most accurate are *master gage blocks* (class AA), guaranteed accurate within plus or minus two millionths of an inch per inch of length. All blocks under one inch have the same tolerance as those one inch long. Master gage blocks are reserved for special research and experimental work and as grand

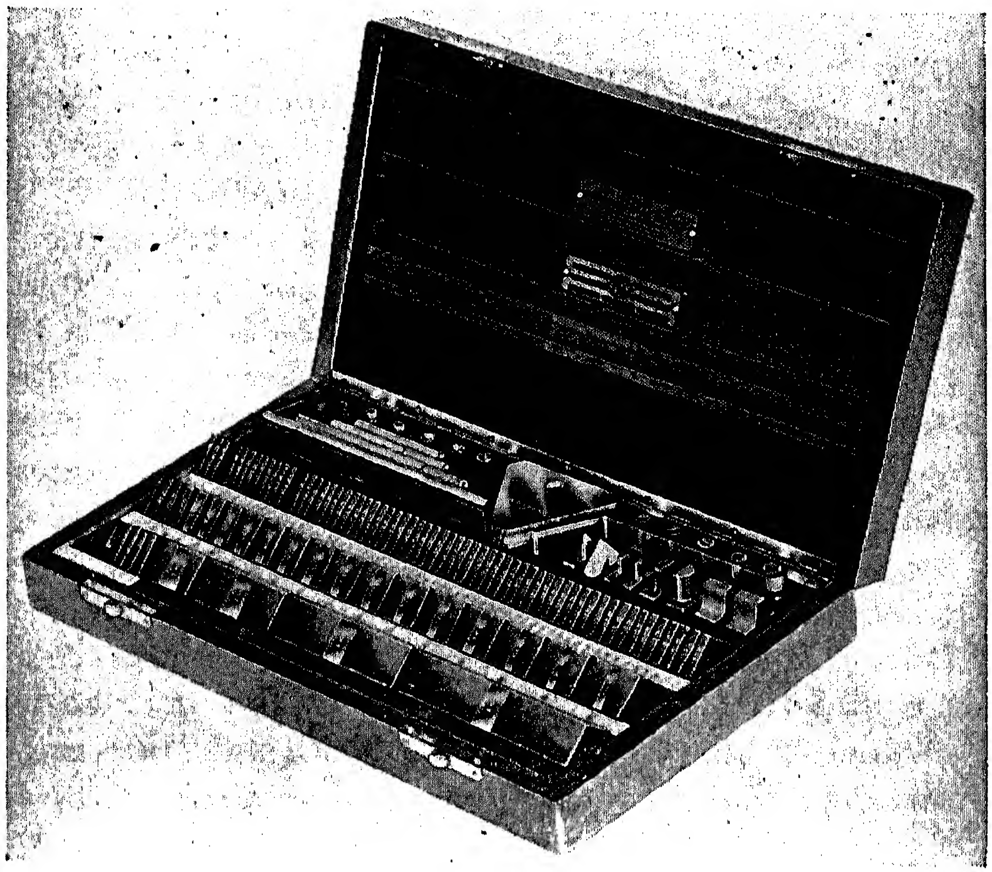


Fig. 3-25. A set of precision gage blocks and accessories. (Pratt and Whitney Photo from Pratt and Whitney Division, Niles-Bement-Pond Co., W. Hartford, Conn.)

masters for checking other gage blocks. Next are the *reference gage blocks* (class A), accurate to plus or minus four millionths of an inch per inch. They are applied to checking working gage blocks, setting gages, calibrating measuring instruments, and doing close layout work. *Working gage blocks* (class B) are held to plus or minus eight millionths of an inch per inch and serve for ordinary layout, inspection, and setting machines and tools. As gage blocks wear, they may be relegated to less accurate work.

Gage blocks are sold individually or in sets ranging from as few as

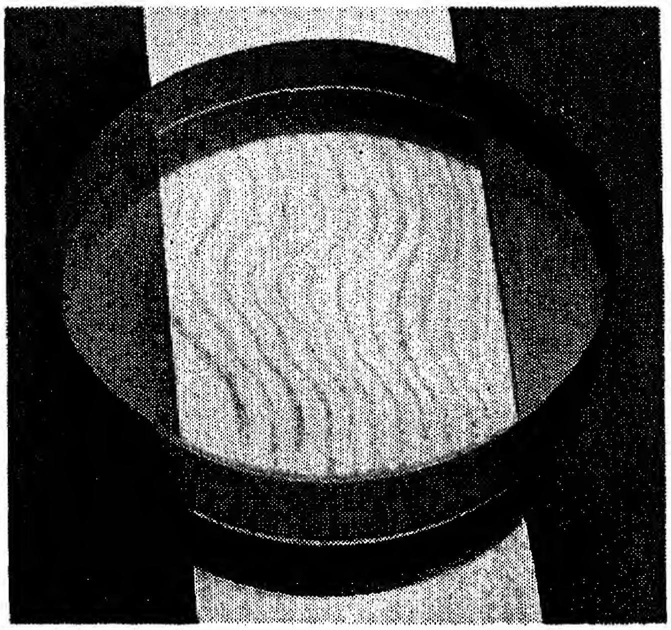


Fig. 3-26. A worn spot detected by an optical flat. (Courtesy The Van Keuren Co.)

5 blocks to as many as 103 pieces. The large sets contain assortments of blocks that can be assembled for measuring many different dimensions. For instance, an 88 piece set gives measurements from 0.200 in. to over 10 in. in increments of 0.000025 in. A large set is shown in Fig. 3-25. In addition to plain blocks, the set contains blocks with special projections. These blocks are assembled with plain blocks to make internal and external measurements, scribe lines, and mark centers. Gage blocks with a square cross section have holes through their centers so that they may be assembled together with tie rods. The square shape is convenient to handle and provides relatively large wearing surfaces. Some gage blocks are narrow, rectangular instead of square, and can be inserted in small spaces. Gage blocks are used to set the instruments in Figs. 3-11 and 3-33.

Optical flat. An optical flat is a clear fused quartz disk 2 to 10 in. in diameter and ½ to ¾ in. thick with faces very near to true planes. The faces of master flats are within 0.000001 in. (one microinch) of absolute flatness. The deviation of less accurate flats may be as much as 0.00001 in.

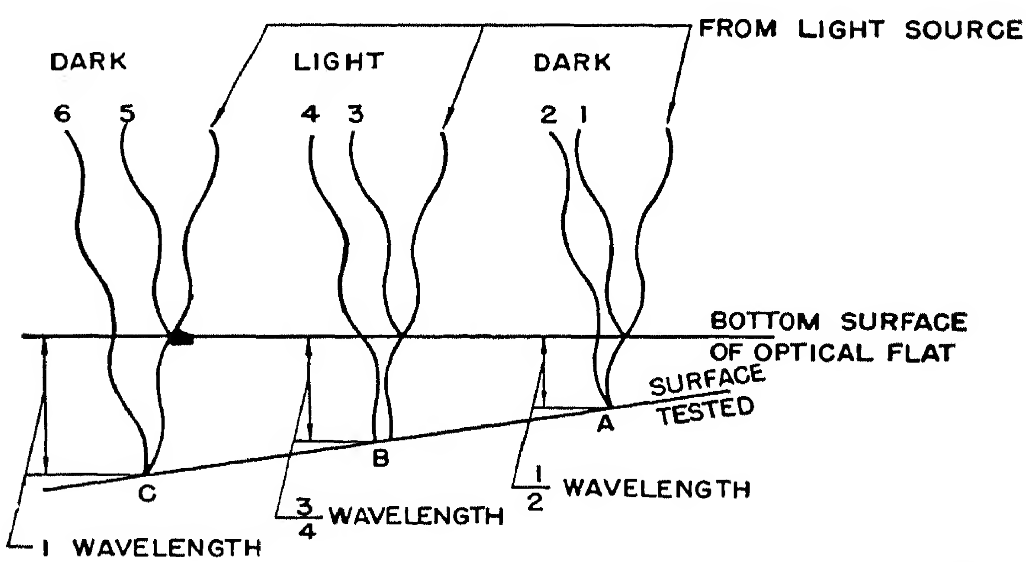


Fig. 3-27. An explanation of lightwave interference.

One use of optical flats is to test the flatness of surfaces. When a flat is placed on a surface to be tested, black bands appear under a monochromatic light wherever the surface is not parallel to the surface of the flat. The appearance of these bands under an optical flat is shown in Fig. 3-26.

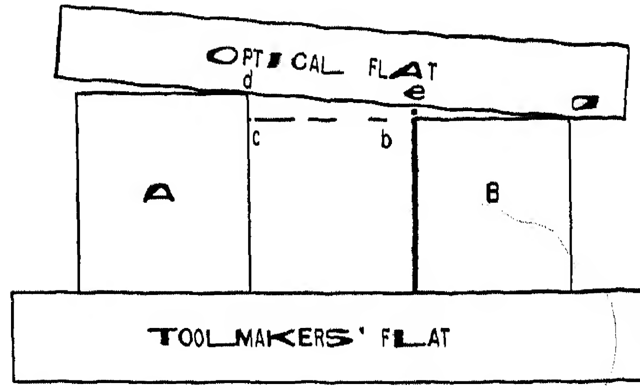


Fig. 3-28. Checking gage blocks with an optical flat.

The explanation of the bands under an optical flat is given by Fig. 3-27. Points A, B, and C are spots on a metallic surface ½, ¾, and 1 wavelength, respectively, below the bottom surface of an optical flat. Light rays of single wavelength are reflected from the bottom of the flat along paths 1, 3, and 5. Some of the light is reflected from the metallic surface along paths 2, 4, and 6. Part of the ray at A travels ½ wavelength through the gap, loses ½ wavelength on being reflected from the dense surface at A, and returns a distance of ½ wavelength. At that stage ray 2 lags 1½ wavelengths behind ray 1, and the two are out of phase and cancel each other. The spot A appears dark. Spot C is also dark because ray 6 has fallen 2½ wavelengths behind ray 5. At spot B, in between, ray 4 has fallen 2 wavelengths (¾ + ½ + ¾) behind ray 3, and the two are in phase. Spot B is light. The vertical distance between the two dark spots A and C is 1 − ½ = ½ wavelength. The wavelength of monochromatic yellow light commonly used is 23.13 microinches, and the vertical distance from band to band is 11.6 microinches.

Optical flats provide means for comparing master gage blocks with others. The way that is done is illustrated by Fig. 3-28. An optical flat is placed on two blocks, A and B, resting on a toolmaker's flat. Block A is higher than block B by an amount *cd*. The flat rests on edges *a* and *d*, and its elevation *be* can be determined by counting the bands over the width *ab*. Then *ac* is measured, and *cd* determined from similar triangles. The same method is used to check other objects, such as balls and rolls, with gage blocks.

Surface Finish Measurements

The elements of surface finish are described in Chapter 2. The purpose now is to show how the characteristics of waviness and

roughness are measured. Waviness can normally be detected by sensitive dial indicators. Roughness is evaluated in a number of ways.

A simple way of appraising roughness is to rub a thumbnail at the rate of about 1 in. per second over a surface. Experienced workmen can estimate roughness to a surprising degree of accuracy by

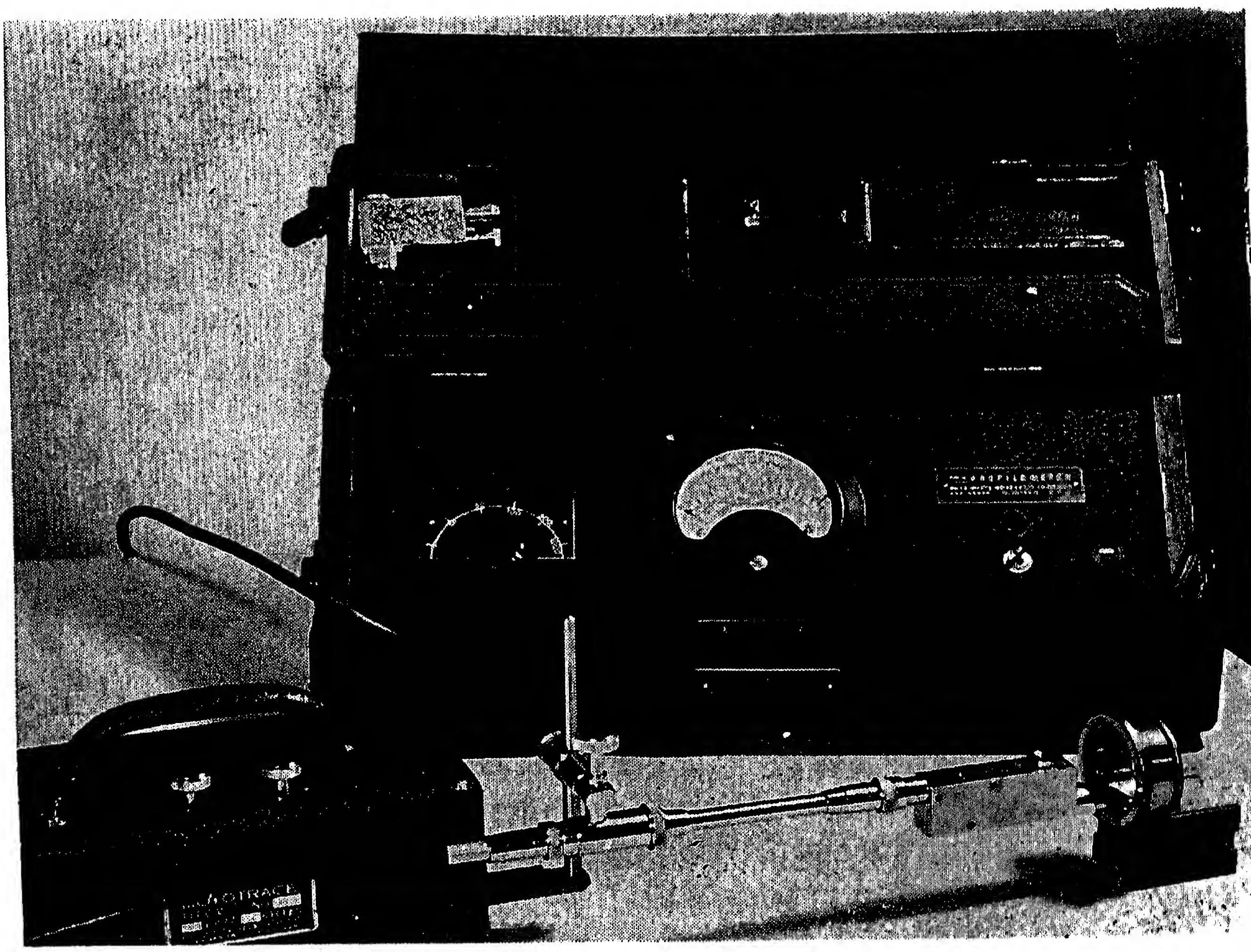


Fig. 3-29. A Profilometer set up for measuring the internal surface of a bearing race. (Courtesy Physicists Research Co.)

this method, but much depends upon judgment, and the method cannot actually be said to measure roughness.

Measurements of roughness are made in two general ways. One is by comparison of surfaces to reference surfaces or discernible standards. The other is by the use of instruments that make direct measurements.

Surfaces may be observed through a magnifying glass or microscope. Standard specimens are commonly used as a basis for visual comparisons. Magnified surfaces are sometimes photographed. By one method, an impression of a surface is made on a transparent

slide; the image is projected and enlarged. Most comparative methods require skill or instruments unsuitable for factory conditions.

Profilometer. The Profilometer shown in Fig. 3-29 is an instrument that gives actual quantitative measurements of surface roughness. A sharp diamond stylus or pointer on the end of an arm is dragged back and forth across a surface at a constant rate. The length of stroke is adjustable. As the stylus passes over the surface, it rises and falls at a rate depending upon the surface roughness. Those movements set up electrical impulses in a coil, and they are amplified and energize a meter. A reading on the meter represents the root-mean-square (rms) average in microinches of the deviations of the surface about a mean plane.

Brush Surface Analyzer. The Brush Surface Analyzer also employs a stylus that explores a surface. The impulses set up by the tracer are amplified and move a pen that records the surface profile, greatly magnified, on a moving tape. A meter may also be connected to the instrument.

Gages

A gage is a device for determining whether one or more dimensions of a manufactured part are within specified limits. When a dimension is gaged, the aim is not to find its actual size but rather to determine whether it lies in a certain range and is acceptable. A clear distinction between measuring instruments and gages is not always observed. Some tools that are called gages are used largely for measuring or layout work. Even some that are used principally for gaging give definite measurements.

According to accuracy, purpose, and use, gages may be classified as working, inspection, and reference or master gages. A *working gage* is used by an operator on a machine to check the work he is producing. *Inspection gages* are used to check finished pieces. *Reference* or *master gages* are used only for checking the size or condition of other gages. A working gage checks to limits inside those checked by the inspection gage. Both are usually inside the workpiece dimension limits. The inspection gage is made with less tolerance than the working gage. A master gage is made as nearly as possible to an exact size.

Certain names are given to gages to describe their features. A gage that establishes the high and low limits of a dimension is called a *limit gage* or a "go" and "not go" gage. Two kinds of limit gages are *progressive* and *double end* gages. The progressive gage has "go" and "not go" members next to each other and is applied to a workpiece with one movement. The "go" member passes into or over a good piece, but the "not go" member does not. A double end gage has the "go" member at one end and the "not go" member at the other. First one end and then the other must be applied to a workpiece. A progressive gage is quicker to use, but its action is sometimes limited. For instance, a progressive plug gage is not suitable for probing the full depth of a blind hole. If the "go" section is long enough to reach the bottom of the hole, it precludes any trial of the "not go" member.

Some gages are fixed for only one set of limits and are said to be *solid gages.* Others are *adjustable* for various ranges. A *receiving gage* has a noncircular hole to receive a part of the proper size and shape. Some gages are considered *standard;* others *special.* Standard gages are produced and sold commercially; special gages are generally made to order. A gage that checks several dimensions at once is called a *combination gage.* A *functional gage* checks the ability of a part to fit or function as intended.

A gage usually is named for a distinguishing feature of its form, shape, method of operation, or application. Common types of gages are plug, ring, snap, feeler, thread, form, and flush pin gages. Those gages make comparisons with the actual dimensions being checked. Other types magnify dimensions or their variations by electric, air, or optical devices. The principal types of gages will be described.

Plug gages. Plug gages are made for checking holes of many forms and shapes – straight, tapered, threaded, round, square, with or without keyways, or splined. Typical plug gages are illustrated on the right-hand side and left center of Fig. 3-30. A straight plug gage often has two diameters for a particular set of limits. Some such gages are progressive, others are double end gages. The gaging members are commonly attached to a handle. The longer is the "go" member and should enter an acceptable hole without being forced. The "not go" member is shorter and should not enter a hole

that is not too large. A tapered plug gage has one member and is marked to show how far it should enter a hole.

An *annular plug gage* is a shell-type plug gage for sizes above

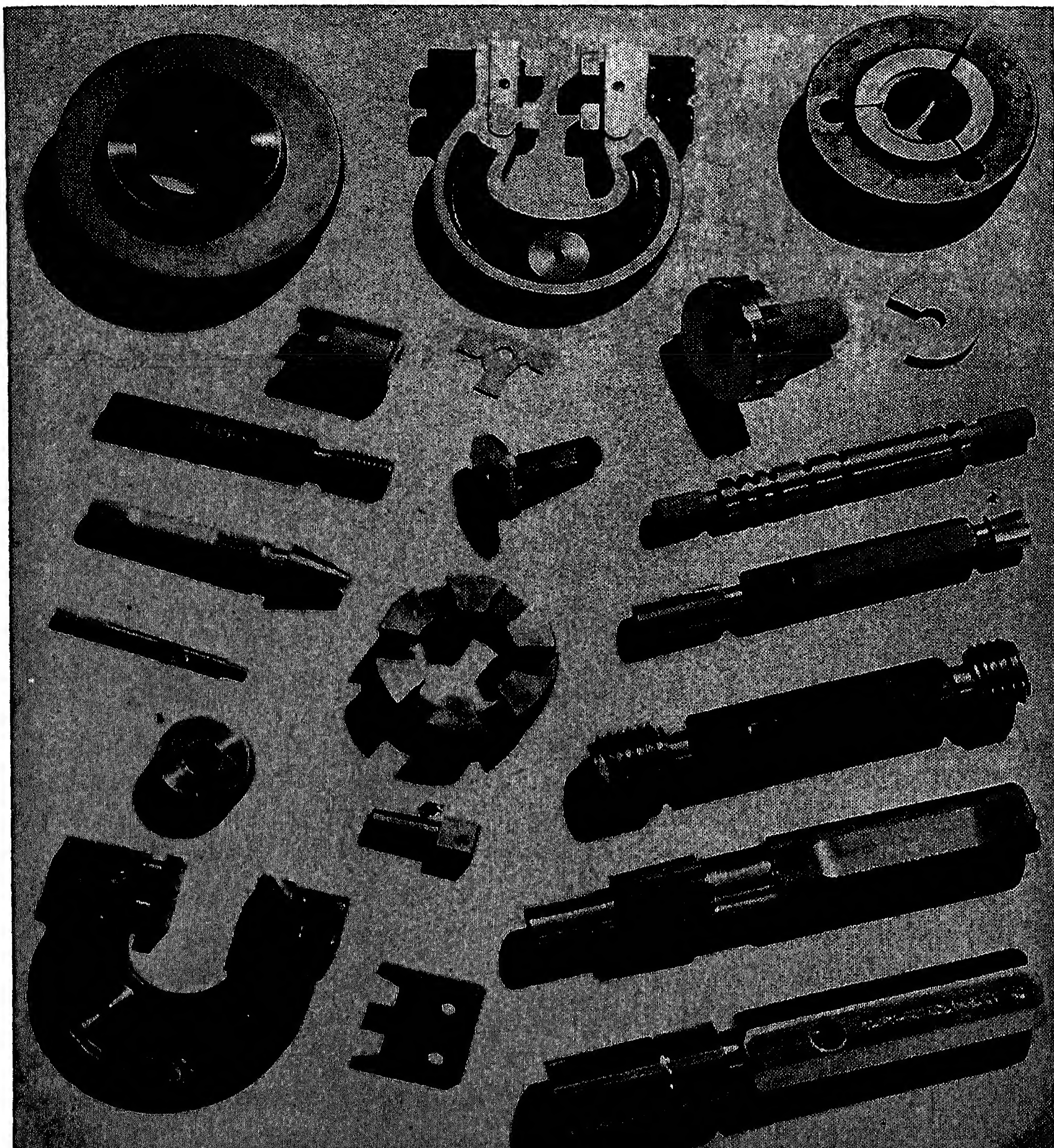


Fig. 3-30. Typical gages. (Courtesy Taft Pierce Co., Woonsocket, R. I.)

8 in. diameter. The inside is machined away to reduce weight, and ball handles are provided for handling. A *flat plug gage* is made in the form of a diametral section of a cylinder. Flat plug gages also are found in large sizes.

Plug and ring gages are made in several classes. Each class calls for a certain degree of accuracy, depending on the size. The smaller the tolerance to be gaged, the smaller must be the tolerance of the gage; but the more accurate the gage, the higher its cost.

Ring gages. A *plain ring gage* tests round straight pieces. One is shown at the top left of Fig. 3-30. A "go" ring gage has a plain knurled outer surface; a "not go" ring gage has an annular groove around the knurled surface. Ring gages also are made for testing tapered and threaded pieces. A *thread ring gage* is shown at the top right of Fig. 3-30. It is adjustable for wear.

Snap gages. Snap gages are caliper-type gages used for checking outside diameters, lengths, and thicknesses. They may be either

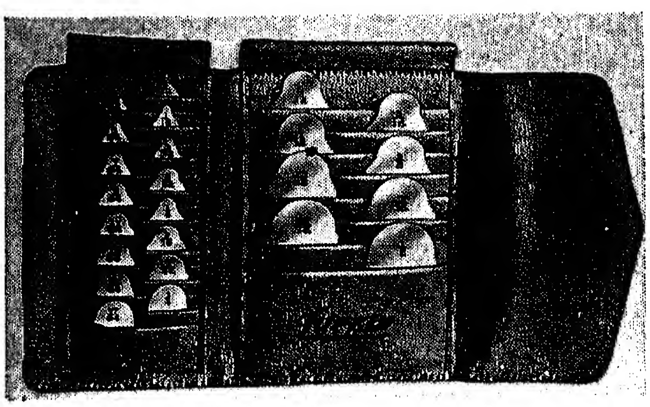


Fig. 3-31. A set of radius gages. (Courtesy The Lufkin Rule Co.)

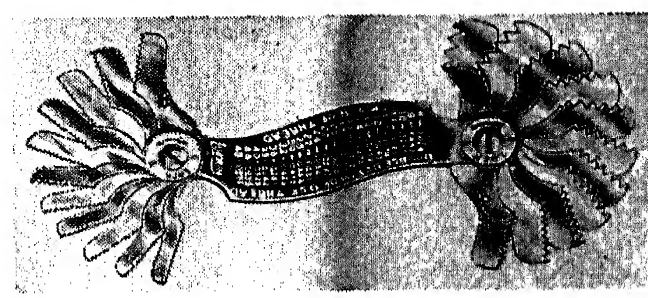


Fig. 3-32. A set of screw thread pitch gages. (Courtesy The Lufkin Rule Co.)

solid or adjustable and progressive or double end gages. Most common are the adjustable progressive snap gages illustrated at the top center and lower left of Fig. 3-30. The snap gage at the top has two sets of wear resistant adjustable buttons. The outer buttons are set to the "go" size, and the inner ones to the "not go" size with gage blocks. The lower gage has a fixed anvil on one side but two adjustable buttons on the other side. Snap gages are also fitted with special anvils, buttons, or rolls for checking external threads and other forms.

Feeler gages. Feeler gages are used to check the widths of slots and other openings and to set cutting tools. They consist of metal strips of definite thicknesses. They often are found in sets consisting of a number of strips ranging in size from 0.0015 in. to about 0.030 in., joined together with a loose joint at one end.

Form gages. An outline of a workpiece may be checked by comparing it to a master contour or shape called a form gage. Form gages made from sheet steel are called *profile* or *template gages*. Several are shown in Fig. 3-30. A profile gage may contain two outlines that represent the limits within which a profile must lie.

Form gages for standard sizes of radii and threads are useful and are commercially available in sets. A set of radius gages is shown in Fig. 3-31. The screw thread gages of Fig. 3-32 serve for roughly checking screw thread pitches.

Flush pin gages. A flush pin gage consists of a body having one or more through holes with a pin in each hole. A surface on the body of the flush pin gage contacts a workpiece surface at one end of the dimension to be gaged. The pin to gage that dimension touches the workpiece surface at the other end of the dimension. The other end of the pin extends to a face of the gage body. A step equal to the tolerance to be gaged is ground on the end of the pin or on the face. If a person puts his finger on the end of the pin and gage body face, he can feel whether the dimension is inside or outside of limits.

Comparators

A comparator is a bench-type instrument employing a means of magnification to show how much a dimension differs from an ideal. Indicators are in this class. The instruments known as comparators are used to check, set, and correlate other gages and tools, and also act as gages themselves. They are true precision instruments and frequently are depended upon to detect differences as small as 0.00001 in. Special comparators with multiple stations are made for production inspection to gage several dimensions at one time. They provide rapid means of inspecting articles made in large quantities like automobile parts and munitions.

Mechanical comparators. Mechanical comparators operate essentially on the same principle as dial indicators. A base with an anvil carries a gaging head on a column. When a workpiece is placed on the anvil and moves the spindle in the head, the amount of movement is magnified mechanically and is shown by a sizable rotation of a pointer on a dial scale. Mechanical comparators are used mostly for inspection of small parts machined to close limits.

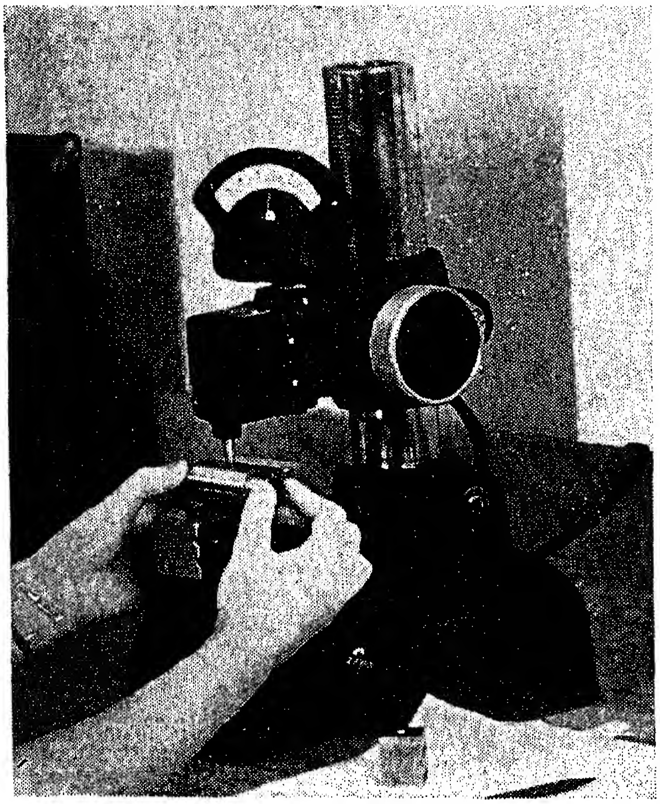


Fig. 3-33. An electric comparator. (Pratt and Whitney Photo from Pratt and Whitney Division, Niles-Bement-Pond Co., W. Hartford, Conn.)

Electric comparators. Electric comparators operate in several ways. One kind, called an *Electrolimit Gage*, is shown in Fig. 3-33. An object to be checked is placed on the anvil on the base under the overhanging gaging spindle. Movement of the gaging spindle unbalances an electric circuit. The displacement is magnified electrically and shown by the hand of a meter. Differences in workpiece sizes can be read to 0.0001 or 0.00001 in. The device can be adjusted for a range of dimensions by raising or lowering the gage head over a distance of several inches. Similar comparators are available for checking internal or height dimensions.

Another kind of electrical comparator has a gaging spindle that moves flexible metal reeds. A needle swings from the reeds and makes and breaks electrical contacts at the limits of its swing. The contacts control signal lights that show when a dimension is outside limits. A red light indicates a dimension below limits, a green light above limits. When a dimension is within limits, no light shows.

Air comparators. An air comparator or *air gage* indicates size by measuring the rate of escape of air in a space between orifices in a gaging head and the surface of the workpiece. A typical air gage with a head for gaging a hole is illustrated in Fig. 3-34. The head on the end of a hose attached to the gage is inserted in a workpiece. For a certain size hole, the clearance between gage head and hole allows air to escape at a definite rate. The rate of air flow is reflected by the height of a column of liquid in a tube on the face of the gage. A small float in the tube defines the liquid level. The gage is set with master ring gages, and pointers are adjusted on the scale to designate the limits. Different gaging heads are used for internal and external dimensions and different sizes.

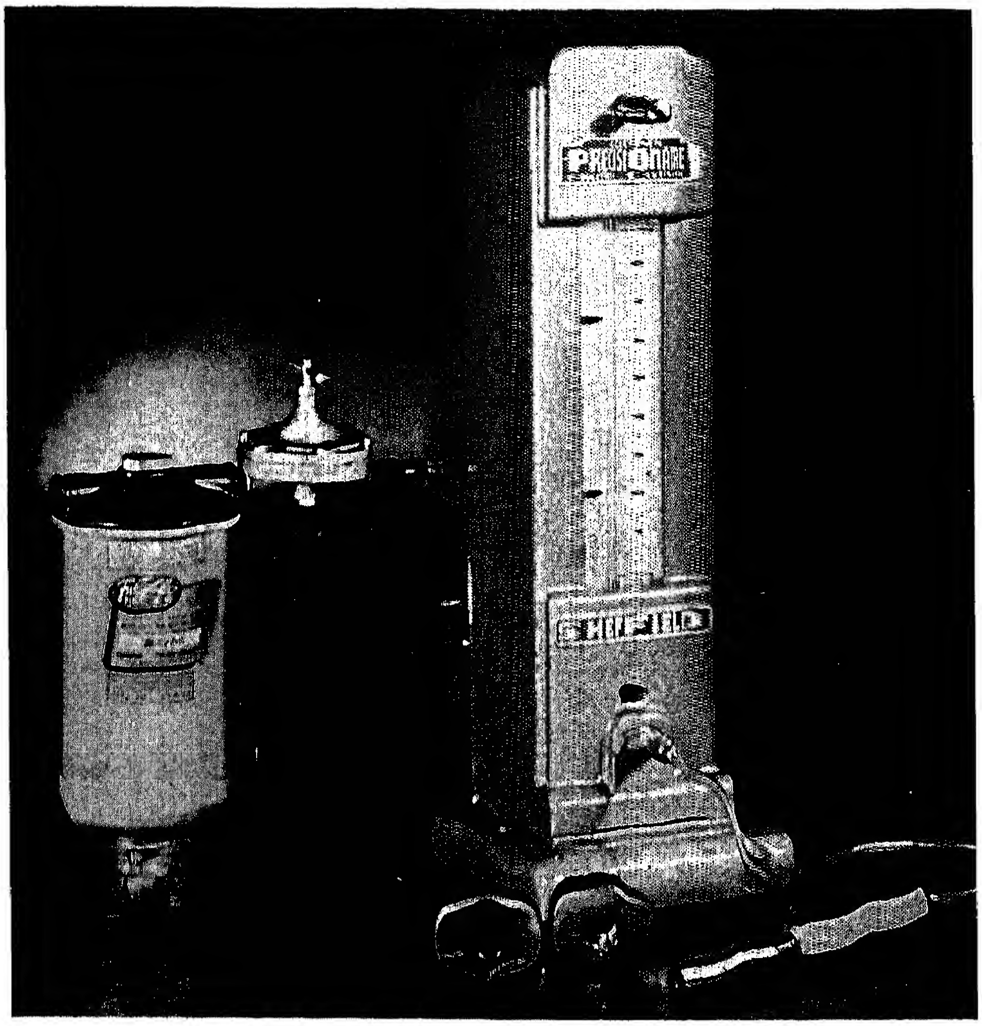


Fig. 3-34. An air gage. (Courtesy The Sheffield Corp.)

Air gages can be applied to measure or gage most kinds of linear and geometric dimensions of holes and outside surfaces. An air gage is often more rapid to use than conventional plug or snap gages. Only one reading is needed for each dimension. The gaging head for a hole is always made smaller than the least size and can be inserted easily. A gage head outwears solid gages 10 to 40 times because it does not fit tightly and the air gage can be readjusted from time to time as wear does take place. Soft surfaces can be gaged without being scratched or marred because the gage head does not have to touch them.

Optical comparators. Optics are employed in a number of ways for measuring and gaging. Various forms of microscopes are widely used for making exacting mechanical measurements. One form is the *toolmaker microscope* that shows objects in their natural rather than inverted positions and is designed for rapid and sure control. Workpieces are held on a table or stage that can be moved and

adjusted by precision screws in two directions and set by gage blocks. A protractor and crosslines are placed in the field of vision. The toolmaker microscope is accurate but rugged and is used in laboratory or shop to check such pieces as screws, gages, tools, jigs, fixtures, and dies.

A *mechanical-optical* or *reed comparator* resembles a mechanical

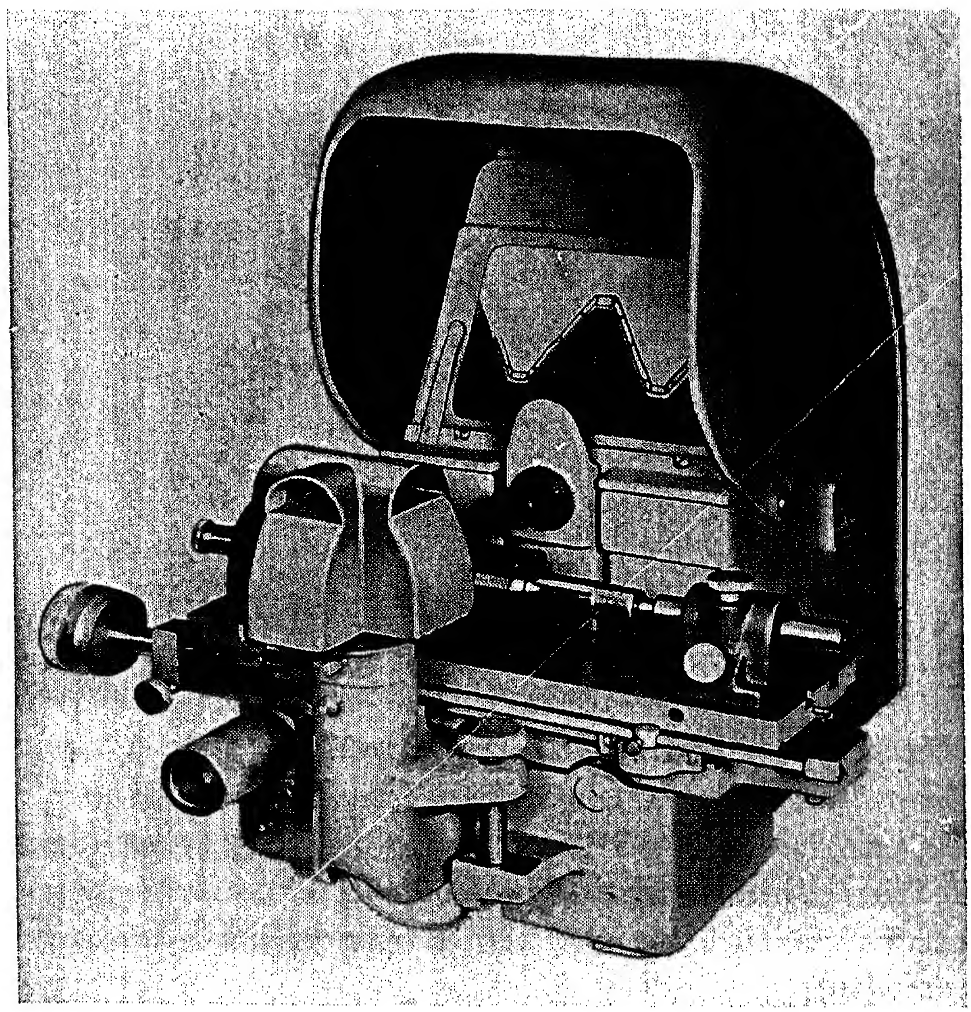


Fig. 3-35. A screw thread shadow cast on the screen of a projection comparator. (Courtesy Jones and Lamson Machine Co.)

or electrical comparator in appearance but magnifies by optical means. Movement of the gaging spindle deflects flexible metal reeds that in turn cause an arm to swing in a definite way to cast a shadow in a beam of light. The light beam is magnified by a series of lenses, and the shadow falls on a scale to indicate the amount of displacement of the spindle. Magnifications up to 20,000 times are obtained. This device has few mechanical parts to wear.

An *optical* or *projection comparator* is a device for projecting an enlarged shadow of an object on a screen where it is compared to

a master chart or drawing. A view of a projection comparator set up for checking a thread hob is shown in Fig. 3-35. A typical thread chart is mounted over the screen on which the shadow of the thread can be seen.

Figure 3-36 is a diagram of a typical projection comparator. Objects are mounted on standard and special fixtures, centers, parallels, etc. on the table. Light rays from the lamp house are sent by a condensing lens in a parallel beam across an object. A projection lens magnifies the image and casts the rays upon a mirror that reflects them to the screen. Interchangeable projection lenses give accurate magnifications from 5 to 125 times.

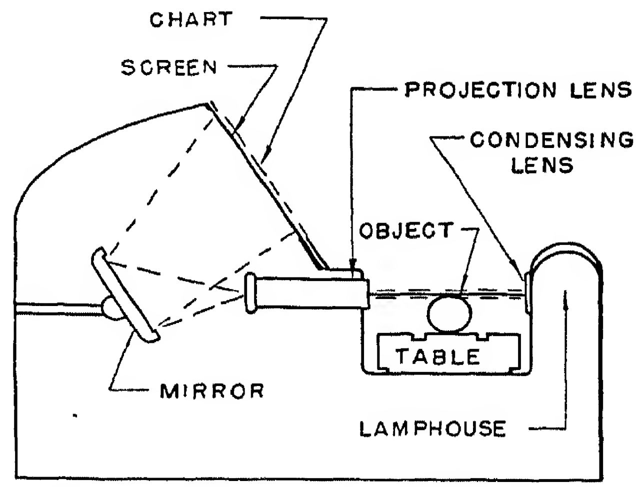


Fig. 3-36. A diagram of an optical comparator.

Questions

1. What are the purposes of measuring instruments and gages?
2. Distinguish between direct and comparative measurements.
3. What precautions should be exercised in using a steel rule?
4. How are calipers used?
5. What measurements can be made with a vernier caliper?
6. What precautions must be taken in using a micrometer caliper?
7. Name and describe 5 instruments using the micrometer principle.
8. What are dial indicator gages and for what are they used?
9. What height of gage blocks should be placed under one end of a 10 in. sine bar to set it to an angle of 30°? 45°? 60°?
10. What does layout mean?
11. What tools are used for laying out work and what do they do?
12. How are surface plates made true?
13. What are gage blocks and what purpose do they serve?
14. Explain the phenomenon of the bands under an optical flat.
15. Draw a diagram and explain how an optical flat and gage block may be used to check closely the diameter of a ball. Of a roll.
16. How may the waviness and roughness of a surface be measured?
17. What are gages and for what are they used?
18. Identify and describe the use of each of the gages of Fig. 3–30.
19. What are comparators?
20. Describe a mechanical comparator and its action, an electrical comparator, and an air comparator.
21. By means of a sketch describe an optical projection comparator.

Problems

1. What are the readings of the vernier scales of (a) Fig. 3-37 A and (b) Fig. 3-37 B, left and right, respectively?

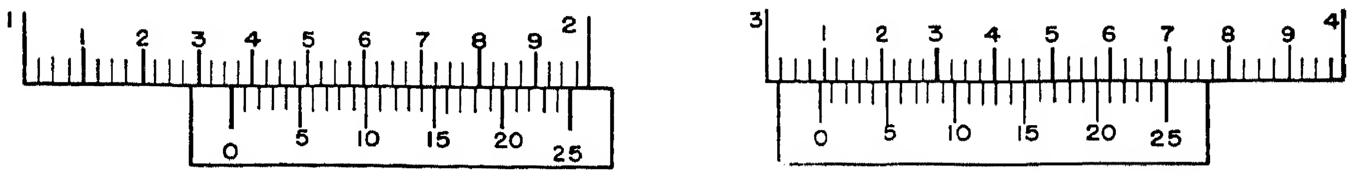


Fig. 3-37. Vernier readings.

2. What are the readings of the micrometer scales of (a) Fig. 3–38 A, (b) Fig. 3-38 B, and (c) Fig. 3-38 C?

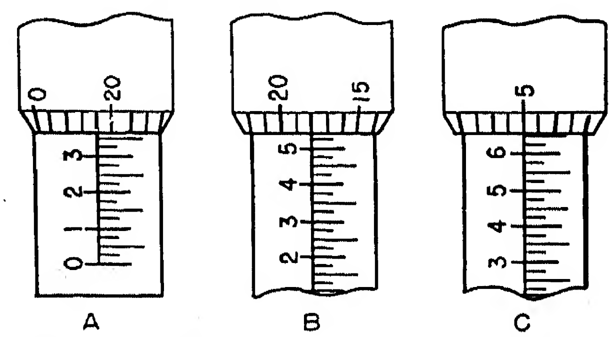


Fig. 3-38. Micrometer readings.

References

Gage Blanks — Commercial Standard CS8–41, Washington, D.C.: U.S. Department of Commerce, 1941.

Michelon, Leno C., *Industrial Inspection Methods,* New York: Harper and Bros., 1950.

Precision Measurement in the Metalworking Industry, Endicott, N.Y.: International Business Machines Corp., 1941.

The Rules and Tools for Precision Measuring, Athol, Mass.: The L. S. Starrett Co., 1947.

Manufacturers' Catalogs:
- Acme Industrial Co., Chicago, Ill.
- Brown and Sharpe Mfg. Co., Providence, R. I.
- Federal Products Corp., Providence, R. I.
- George Scherr Co., New York, N.Y.
- Jones and Lamson Machine Co., Springfield, Vt.
- L. S. Starrett Co., Athol, Mass.
- Lufkin Rule Co., Saginaw, Mich.
- Pratt and Whitney Division, Niles-Bement-Pond Co., Hartford, Conn.
- Standard Gage Co., Poughkeepsie, N.Y.
- The Do All Co., DesPlaines, Ill.
- The Sheffield Corp., Dayton, Ohio.

Chapter 4

CUTTING TOOLS AND THEIR ACTIONS

THE MOST IMPORTANT PART of a metal machining operation is the spot where the cutting tool meets the workpiece and pries away chips. An understanding of what happens in the cutting zone is necessary to appreciate what makes a good cutting tool and how it should be operated.

The Requirements for Efficient Metal Cutting

How cutting tools cut metal. When a tool cuts metal, it is driven by a force necessary to overcome friction and the forces that hold the metal together. The metal that the tool first meets is compressed and caused to flow up the face of the tool. The pressure against the face of the tool and the friction force opposing the metal flow build up to large amounts. Figure 4-1 is a diagram of the action in a single plane of a cutting tool forming a chip. At point A the material may be sheared by the advancing tool or torn by the bending action of the chip to start a crack. The stress in the material ahead of the advancing tool reaches a maximum value in a plane approximately perpendicular to the tool face. That plane is known as the *shear plane,* and one edge is depicted by the line AB of Fig. 4-1. When the strength of the metal is exceeded, rupture or slippage occurs along the shear plane. With further movement, new material is compressed by the tool, and the cycle is repeated again and again. As it travels along, the cutting edge scrapes and helps clean up the surface.

Cutting forces. A force is exerted on the face of a tool by the

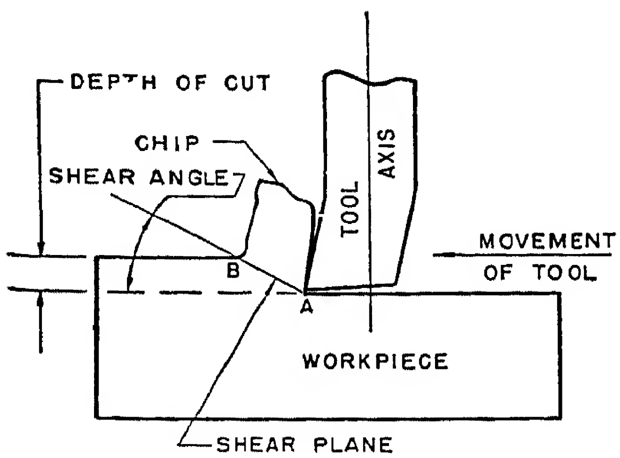


Fig. 4-1. A diagram of the action of a cutting tool.

material pushed ahead, and a friction force is set up along the face of the tool by the sliding chip. These forces have a resultant, and an equal and opposite force must be applied to the tool to make it cut. The driving force on the tool of Fig. 4-1 may be resolved into two components for convenience. One is parallel to the axis of the tool, and the other acts in the direction of the movement of the tool. The second alone determines the power required because it is in line with the tool movement.

The force driving a lathe tool like the one sketched in Fig. 4-7 has three principal components. One is called the *vertical* or *tangential component* because it acts vertically, tangent to the workpiece. The second is the *longitudinal* or *traversing component* and acts parallel to the axis of the workpiece in the direction the tool is moved. The third is called the *radial* or *normal component* and acts radially on the workpiece and normal to the finished surface in a horizontal plane. Both the vertical and longitudinal forces affect the power required to make the cut, but the velocity of the work surface is much greater than the traverse or feed rate of the tool, and most of the power is related to the vertical force.

Many tests have been made to determine cutting forces and power requirements under various conditions. The results are given in reference books and handbooks.

Types of chips. When a brittle material like cast iron or bronze is cut, it is broken along the shear plane. The chips come off in small pieces or segments and are pushed away by the tool as illustrated in the highly magnified view of Fig. 4-2. A chip formed in this way is called a *Type I* or *segmental chip*.

A ductile material, like aluminum, is not broken up but, when cut under favorable conditions, comes off like a ribbon as shown by the highly magnified section of Fig. 4-3. This is known as a *Type II* or *continuous chip*. An evident line of demarcation separates the highly distorted crystals in the chip from the undistorted parent material. That line is the edge of the shear plane at one instant and corre-

sponds to line AB of Fig. 4-1. As the material slips along one plane, it is work-hardened and resists further distortion. The stresses build up on the next plane to bring about slippage in new material, and so on.

When steel is cut, a continuous chip is formed, but the pressure against the tool is high, and the severe action of the chip quickly rubs the natural film from the tool face. The freshly cut chip and the newly exposed material on the face of the tool have an affinity for each other, and a layer of highly com-

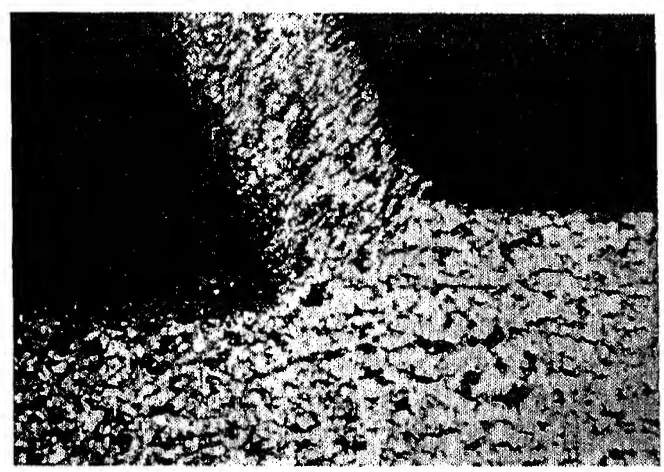


Fig. 4-3. A photomicrograph of a continuous chip. (Courtesy The Cincinnati Milling Machine Co.)

Fig. 4-2. A photomicrograph showing how a chip is formed and ruptured from a brittle material. Successive positions of the advancing tool are depicted from top to bottom. (Courtesy The Cincinnati Milling Machine Co.)

pressed material adheres to the face of the tool. That formation is illustrated in Fig. 4-4 and is called a *built-up edge.* The chip is a *Type III* or *continuous chip with built-up edge.* As the cut progresses, the pile on the face of the tool becomes large and unstable. At frequent intervals pieces topple from the pile and adhere to the work surface or pass off with the chip. The build-up edge pushes on ahead of the tool and to a certain extent protects its edge.

The criteria for judging cutting tool performance. Four considerations are important for judging how well a cutting tool performs. They are (1) the rate at which metal is removed, (2) the length of time the tool lasts, (3) the power consumed in making the cut, and (4) the surface finish and accuracy obtained.

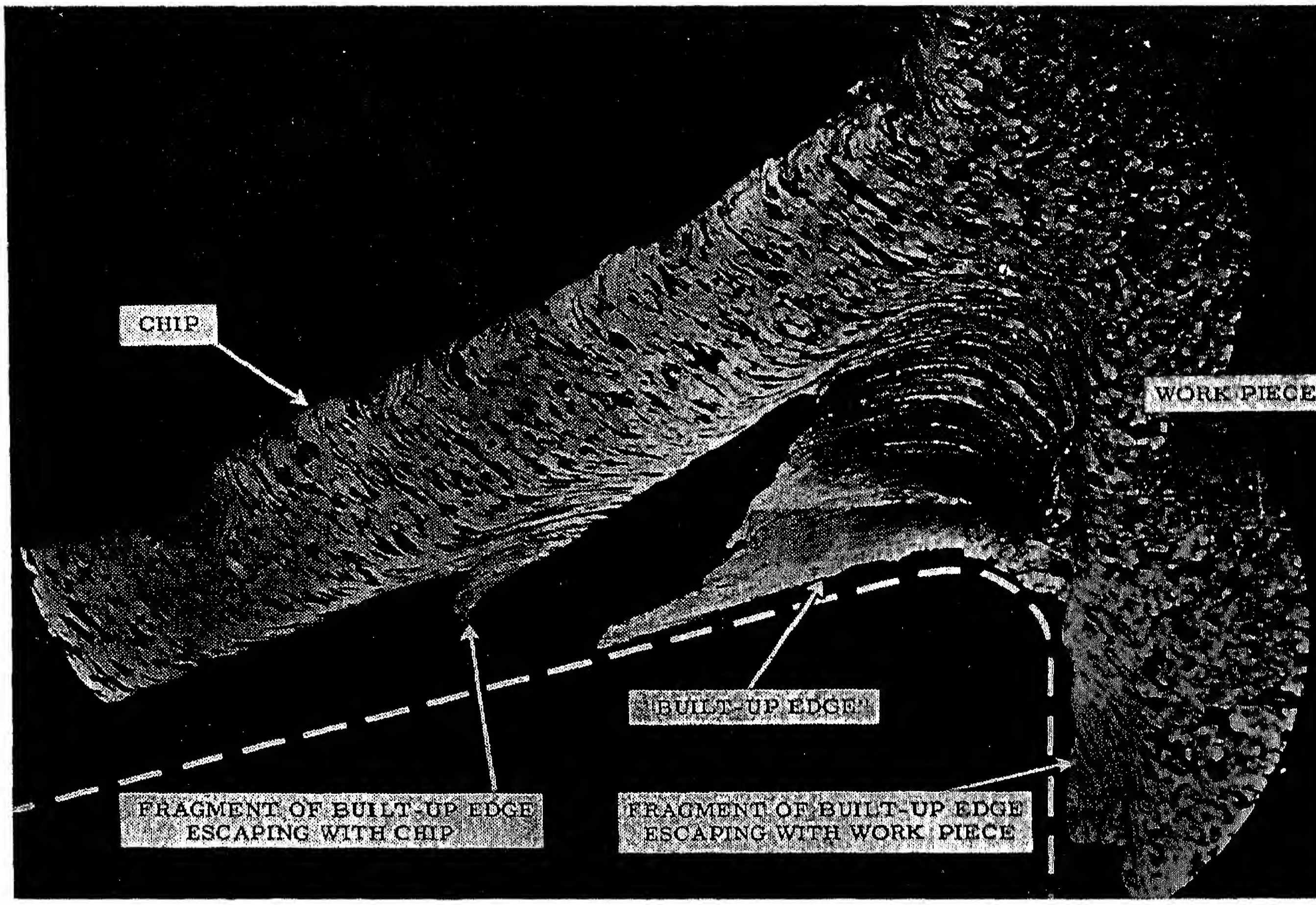


Fig. 4-4. A composite photograph of a highly magnified chip and a built up edge. (Courtesy The Cincinnati Milling Machine Co.)

The *rate of stock removal* is defined as the volume of material, in cubic inches, removed in a unit of time, usually a minute. It determines the time required to remove a specified amount of material from a workpiece. This is an important consideration because, as has been estimated, half of the material is removed from an average rough workpiece to produce a finished part. In some cases the quantity removed may be as high as 80 or 90 per cent. Metal must obviously be removed rapidly, if machining is to be economical.

The *length of time a tool lasts* or cuts satisfactorily is called the *tool life*. After a tool has been cutting for some time, depending upon conditions, its edge breaks down. Work material or chips rubbing over the tool may wear a crater on the flank below the cutting edge, on the face behind the edge, or on both. As the crater is enlarged, the tool material supporting the edge is removed. Without support, the edge breaks off eventually, and the tool fails. Even before a crater is formed, the edge of a tool may chip off or crumble when penetrating a hard material or meeting hard spots in a soft

material. Too high a cutting temperature, which may result from too high a cutting speed, softens and ruins a tool edge. When the edge breaks down, cutting forces and power consumption rise rapidly, and the tool ceases to cut cleanly.

A tool should be removed and reground before complete failure occurs. If the tool is allowed to deteriorate too far, an excessive amount of material must be ground off to recondition it. If a tool is too weak or is used improperly, it may break before the cutting edge becomes dull. After a tool has been reground a number of times or broken, it must be reworked or replaced. Tool material and the labor for sharpening, resetting, readjusting, and reworking tools make up a definite part of the cost of any machining operation. These costs must be kept low, consistent with reasonable rates of metal removal, if machining is to be economical.

The *power consumed* in driving a cutting tool is proportional to the force applied to the tool times the velocity of the tool moving through the material. At a constant velocity, the power required is directly proportional to the force in the direction of movement, and that force is related to the force of friction and the force that causes rupture or shear along the shear plane. The magnitude of the friction force between the chip and tool depends upon the coefficient of friction. Conditions that reduce the coefficient of friction decrease the friction and the power consumption. The force to rupture or displace the material along the shear plane depends upon the area of the plane. The larger the shear angle depicted in Fig. 4-1, the smaller the area of the shear plane for a constant depth and width of cut. Various things can be done to keep down the friction and shearing forces in metal cutting. The important ones will be described later in this chapter.

Most of the power used in metal cutting is dissipated in the form of heat. Thus, less power consumption means less heat and lower temperatures, and that is favorable to longer tool life.

The *surface finish and accuracy* obtained on a job must be good enough to satisfy specifications but are uneconomical if much better than required. A roughing cut that is to be followed by a finishing cut on a surface generally need not produce a smooth finish nor hold a close tolerance. However, a finishing cut often is expected to leave as good a surface as the process is capable of producing and to maintain small tolerances. The surface finish and accuracy pro-

duced in a cut depend upon the condition of the edge of the tool, the built-up edge, and vibration.

Except where a built-up edge is present, the tool edge cleans up and sizes a surface. As the edge wears, the size changes. If a small tolerance must be held, tool edge wear may necessitate a number of adjustments during the life of the tool. The edge of a tool leaves its imprint upon the surface it cuts. For instance, a tool with a sharp point leaves a groove in its path.

As shown in Fig. 4-4, pieces of built-up edge slough off during a cut, pass under the tool, and adhere to the work surface at frequent intervals. The surface is left covered with a multitude of the fragments of the built-up edge. These fragments largely constitute the roughness of a machined surface. Factors that deter the formation of a built-up edge improve surface finish.

Metal cutting is cyclic. The cutting force builds up to cast off a chip segment or cause slip in the shear plane. Then momentarily the force may drop off while the tool is taking another bite, and the cycle is repeated. These changes occur at high frequencies and together with other interruptions in the cut often induce vibrations in machine members, tools, and workpieces that are not well supported. For instance, a weak and flexible tool may deflect and dig into the work when the force is high and then snap back when the force decreases. Excessive vibration in machining operations is known as *chatter*. It may be quite noisy and obnoxious, can cause damage to machine tools, and defaces work surfaces with patterns called chatter marks.

The factors that affect metal cutting efficiency. The degree of surface quality and accuracy that a cutting tool must produce is generally specified. The rate of metal removal, tool life, and power consumption should be such that the specifications are met at the lowest total cost per piece. Performance in each of these areas depends upon factors that can be varied to produce desirable results These factors are:

1. the tool material
2. the shape and form of the tool
3. the work material
4. the cutting speed
5. the feed and depth of cut
6. the use of cutting fluid

These factors and the way each influences performance will now be described.

Cutting Tool Material

Hardness is the first requisite of a cutting tool material because it must be able to penetrate other materials. Toughness is also desirable to withstand shock. Cutting tools must work upon many kinds of metals and under a variety of conditions. No one cutting material is the best for all purposes. The principal cutting tool materials are:

1. carbon tool steel
2. high speed steel
3. cast nonferrous alloys
4. cemented carbides
5. diamonds
6. artificial abrasives (described in Chapter 18)

Carbon tool steel. Carbon tool steel is the oldest kind of cutting material but is little used today compared with other kinds. It contains from 0.90 to 1.20 per cent carbon and very little alloying elements. The chief disadvantage of carbon tool steel is that it softens at temperatures above 400° F. and therefore is limited to very slow cutting speeds and light duty. However, at low temperatures carbon tool steel is hard, wear resistant, and as serviceable as more expensive materials for some applications. It is comparatively cheap and suitable for special tools, like odd sizes of drills, that are infrequently and lightly used and do not warrant much expense. It is easy to fabricate and simple to harden.

High speed steel. High speed steel is so named because it cuts at higher speeds than other steels. Even so, it is limited to speeds and conditions that do not give rise to a temperature above about 1100° F. At red heat, high speed steel loses its hardness and breaks down quickly. It is moderate in cost and tough, which makes it preferable in many cases to harder but more brittle and expensive materials. For instance, high speed steel often is able to withstand interrupted cuts better than harder materials. High speed steel can be made into various forms of tools with reasonable facility but requires care in its heat treatment.

High speed steel may contain tungsten, molybdenum, chromium,

vanadium, and cobalt. These elements are alloyed in various proportions in different types and grades of high speed steel. One of the oldest and most common types of high speed steel is known as 18-4-1. It contains about 0.55 to 0.75 per cent carbon, 18 per cent tungsten, 4 per cent chromium, and 1 per cent vanadium. Molybdenum is used in some types to eliminate partially or wholly the need for tungsten. A general-purpose and widely used "moly" high speed steel contains 8 per cent molybdenum, 4 per cent chromium, 1½ per cent tungsten, and 1 per cent vanadium. Its performance is comparable to tungsten high speed steel. Cobalt is sometimes added to high speed steel to improve red hardness. Vanadium also acts to increase the hardness of high speed steel.

Cast nonferrous alloys. Cast nonferrous alloys contain no iron and cannot be softened by heat treatment so as to be machined easily. They must be cast to shape and ground to size. A typical cast nonferrous alloy has the trade name of *Stellite* and contains 43 to 48 per cent cobalt, 17 to 19 per cent tungsten, 30 to 35 per cent chromium, and about 2 per cent carbon, depending upon the grade. Different grades are intended for different purposes. Stellite retains its hardness and becomes tougher at red heat. It has a particularly hard skin that stands the abrasive action of such materials as cast iron, malleable iron, and bronze.

Cemented carbides. Cemented carbides, sintered carbides, or just carbides are composed of very hard particles held together by a metallic bond. The main ingredient is tungsten carbide, but it is often mixed with various amounts of tantalum and titanium carbides to improve resistance to abrasion and lower the coefficient of friction, which is particularly beneficial in cutting steel. The carbide particles are mixed with a binder of nickel or cobalt powder. The mixture is compressed in a mold and presintered. At this stage the material can be and is cut to desired shapes. The blanks are then sintered at high temperatures to form a solid and hard substance. The blanks are then finished by grinding.

Cemented carbides have a high first cost but are economical for machining parts in large quantities because they stand up under high speeds, remove metal rapidly, and give good finishes. Carbides are made in a number of grades by varying the proportions of carbide particles and binder. At one extreme are the hardest but most brittle carbides with high resistance to abrasion and wear.

In other grades hardness is sacrificed in various degrees for strength and shock resistance. Cemented carbides are made and sold under a number of trade names such as *Kennametal* and *Carboloy*.

Diamonds. The diamond is the hardest known material and can be run at cutting speeds up to 5000 surface feet per minute. Diamonds are suitable for cutting very hard materials and producing fine finishes. They are brittle, do not conduct heat well, and are limited to light cuts. A typical application is the precision boring of holes.

Comparison of cutting tool materials. A comparison is made in approximate terms in Table I of the endurance and cost of the four common cutting tool materials. In general, the higher priced and more productive materials are warranted for large-quantity production. However, each material has certain features that make it desirable for specific applications.

Table I

Comparison of Cutting Tool Materials

Material	*Proportionate tool life — per cent* [1]	*Typical prices* [2]
Carbon tool steel	30-50	$ 0.10
High speed steel	100	0.50
Nonferrous alloy	150-200	1.50-2.50
Cemented carbides	300-1000	5.25

1 The life of high speed steel under normal conditions is represented by 100%. The relative lives for cutting tools of the other materials are for comparable conditions.
2 Prices are typical for standard ⅜ in. square tool bits but vary with quantity, grade, and market conditions.

As indicated in Table I, high speed steel may last from two to three times as long as carbon tool steel under comparable conditions. That means that a high speed steel tool will cut for a longer time before it must be reground. In other words, less time is needed for grinding, resetting, and adjusting the high speed tool. Thus the actual productivity of that tool may be from seven to ten times that of a carbon tool steel tool — more than enough to justify the cost ratio of five to one.

A typical report on machining heat treated steel shafts showed 1284 cubic inches of metal removed per grind by cemented carbide tools as compared with 160 cubic inches per grind for high speed steel tools, at a cost of $1.42 per shaft in the first case as against

$3.19 in the second. A carbide tool can turn out more pieces and show a total savings in production of much more than its first cost.

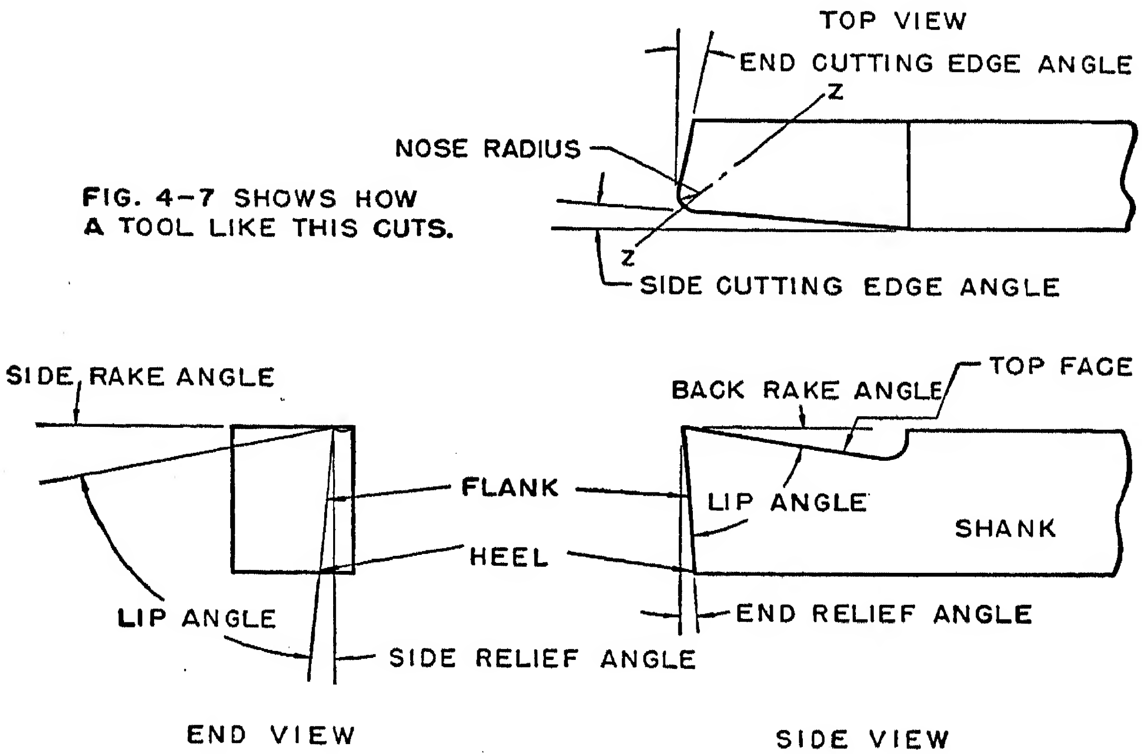


Fig. 4-5. Conventional cutting tool angles.

Cutting Tool Shapes and Forms

Tool angles. The *point* is that part of a cutting tool where cutting edges are found. It is on the end of the *shank* or *body*. The surfaces on the point bear definite relationships to each other that are defined by angles. The angles of a single point cutting tool of a type used on a lathe, shaper, or planer are sketched in Fig. 4-5. The main elements that define the shape of a tool point are the back rake, side rake, end relief, side relief, end cutting edge, and side cutting edge angles and nose radius. The angles are measured in degrees and the radius in inches. The shape of a tool often is designated by a series of numbers specifying the values of the angles and radius in the order just listed. Thus a tool with a shape specified as 8-14-6-6-6-15-3/64 has 8° back rake, 14° side rake, 6° end relief, 6° side relief, 6° end cutting edge, 15° side cutting edge angles, and a 3/64 in. nose radius.

The positions of the angles on a tool are determined by the way the tool acts. The lathe or shaper tool of Fig. 4-5 is designed to

enter the material top first, and the rake angles are on the top face. The slotter tool of Fig. 4-6 travels axially in cutting, and its rake is on the end. As other types of tools are studied, their angles will appear in various positions but will still relate to the ways the tools act. The sizes of the angles largely affect tool performance.

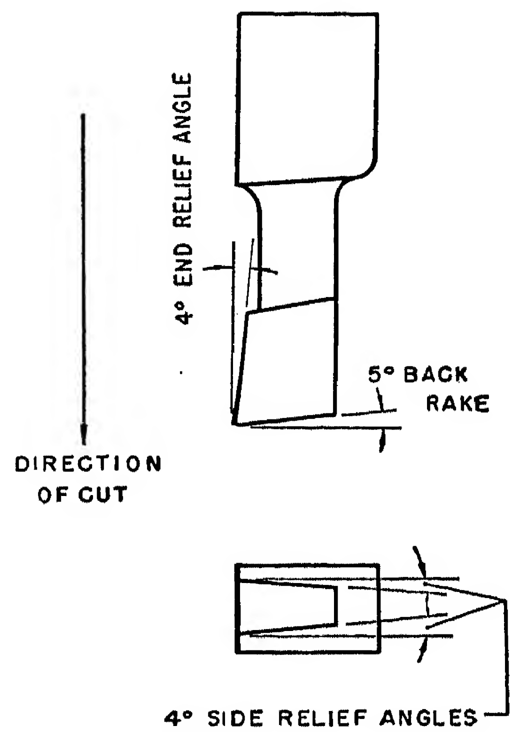


Fig. 4-6. A typical slotter tool.

Relief angles. The purpose of a relief angle, as the name implies, is to enable a surface of a tool to clear the work and not rub. A relief angle lies between a plane perpendicular to the base of a tool and the surface immediately below the cutting edge. The amount of relief needed depends upon the kind of cut. The tool of Fig. 4-7 is engaged in turning and advances as the workpiece revolves. The side relief must be greater than the helix angle of the cut. If a tool is on center, as in Fig. 4-7, the smallest front relief between the tool and work is at the very tip. If the tip of the tool is above center, the effective relief is reduced; below center the effective relief is increased. The amount of relief should not be more than necessary for free cutting because an excess relief angle removes support from the cutting edge and weakens the tool. Relief angles between 5° and 12° are usually ground on lathe tools. Carbide tools commonly have small relief angles for maximum support of their brittle cutting edges. Often a secondary relief angle, called a *clearance angle*, is ground on the shank below the insert of a carbide tool. A shaper or planer tool is subject to repeated shocks as it enters the material at the beginning of its strokes and is given relief angles as small as 4°. In general, relief angles should be small to cut hard materials but may be larger for soft materials.

Rake angles. The rake angle of a tool affects the angle of shear during the formation of a chip. The larger the rake angle, the larger the shear angle and the lower the cutting force. However, increasing the rake angle decreases the cutting angle and leaves less metal

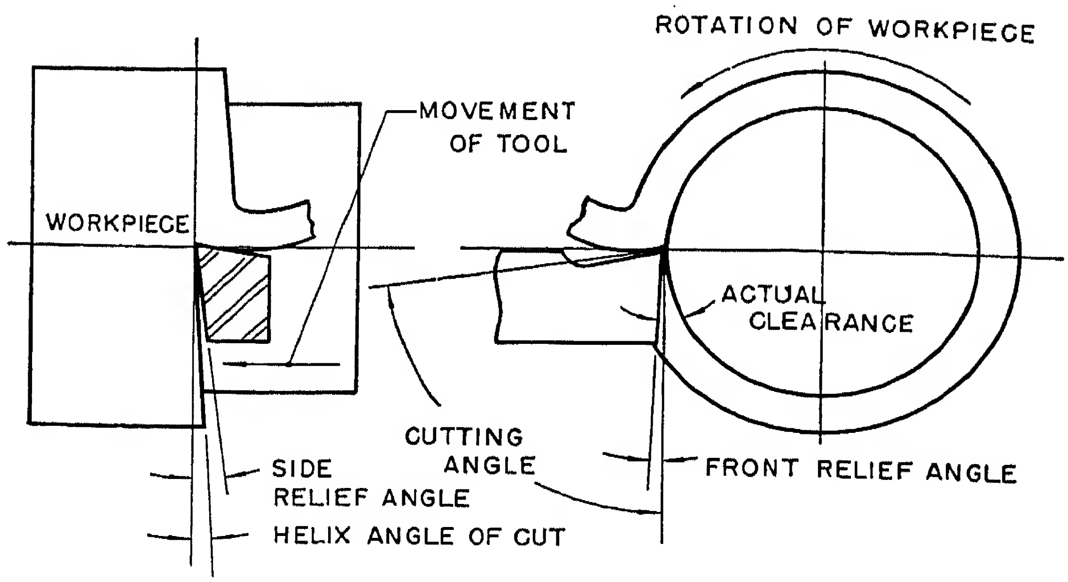


Fig. 4-7. How the clearance angles on a lathe tool prevent rubbing.

at the point of the tool to support the cutting edge and conduct away heat. The smaller the cutting angle, the less the power required for a cut but the sooner the cutting edge is likely to fail. A practical rake angle represents a compromise between these two considerations. In general, the rake angle is small for cutting hard materials and large for soft ductile materials.

The two conventional components of rake are back rake and side rake, designated by the rake angles of Fig. 4-5. The tool in that figure is designed to cut on the side cutting edge, the nose radius, and to some extent on the front cutting edge. The chip separated by the cutting edges flows along a line Z-Z. The true rake angle is the compound angle in the plane through Z-Z normal to the base of the tool and is determined by both the side and back rake angles. Side rake is more important than back rake on side cutting tools for turning and similar operations. A steep side rake angle tends to direct the chip to the side away from the toolholder, reduces side deflection of the tool, cuts down the force in the direction of feed, and weakens the tool less than a steep back rake.

High speed steel tools for cutting hard chilled iron have no rake because full support is needed for the cutting edge to keep it from spalling. For hard steel and cast iron, rake angles from 8° to 14° are desirable; for soft steel, 15° to 25°; and for aluminum, up to 35°. Although brass is a soft material, it is usually cut with a zero or negative rake angle to keep the tool from digging into the work.

Tools of nonferrous cast alloys and cemented carbides are more brittle, need more support for their cutting edges, and are given smaller rake angles than high speed steel tools. Rake angles of 0° to 4° are recommended for cast iron, bronze, and steel; and 10° to 20° for aluminum. The rake angles referred to so far and those shown in Fig. 4-5 are positive. Rake angles measured counterclockwise from zero are called negative rake angles. Negative rake angles have been found to give good results on carbide tools under certain conditions, such as where the tools are subjected to severe shocks from interrupted cuts. Negative back rake angles are usually held within 2° to 10°, accompanied by positive side rake angles about 2° to 4° larger in numerical value than the negative rake angles. A tool with a negative rake receives initial impact behind the cutting edge when it starts to cut, and its edge has added material for support. A negative rake angle does increase the cutting forces at lower speeds, but carbide tools with negative rake can be run at very high speeds at which cutting forces drop off.

Cutting edge angles. An *end cutting edge angle* gives clearance to the trailing end of the cutting edge and reduces drag that tends to cause chatter. Too large an end cutting edge angle takes away metal that supports the point and conducts away heat. An angle of 8° to 15° has been found satisfactory in most cases for side cutting tools, like turning and boring tools. Sometimes a flat 1/16 to 5/16 in. long is ground on the front edge as shown in Fig. 4-8 so that the edge can get in a wiping action to help produce a good finish. End cutting tools, like cut-off and necking tools, often have no end cutting edge angle.

A *side cutting edge* or *lead angle* affects tool life and surface finish. It enables a tool that is fed sideways into a cut to contact the work first behind the tip. A side cutting edge at an angle has more of its length in action during a definite depth of cut than it would without an angle. The result is a wider and thinner chip that distributes the cutting forces and heat over more of the working edge and keeps down the formation of a built-up edge. On the other hand, the larger the angle, the greater the component of force tending to separate the work and tool. That promotes chatter. The most satisfactory side cutting edge angle is generally 15°, although sometimes it is made as large as 30° to 40°. No side

cutting edge angle is desirable when cutting castings or forgings with hard and scaly skins, because then the least amount of side edge should be exposed to the destructive action of the skin.

A *nose radius* is favorable to long tool life and good surface finish. A sharp point on the end of the tool is highly stressed and short lived. A sharp point leaves a groove in the path of cut, and that produces an inferior surface finish. On the other hand, too large a radius is conducive to chatter. Radii from 1/32 in. to 1/4 in. are common. A nose radius from 1/2 to 2/3 the depth of cut is often considered satisfactory. The size of a nose radius may be prescribed by the fillet that must be left in the corner at the end of a cut. A sharp pointed tool is necessary if a sharp corner is required.

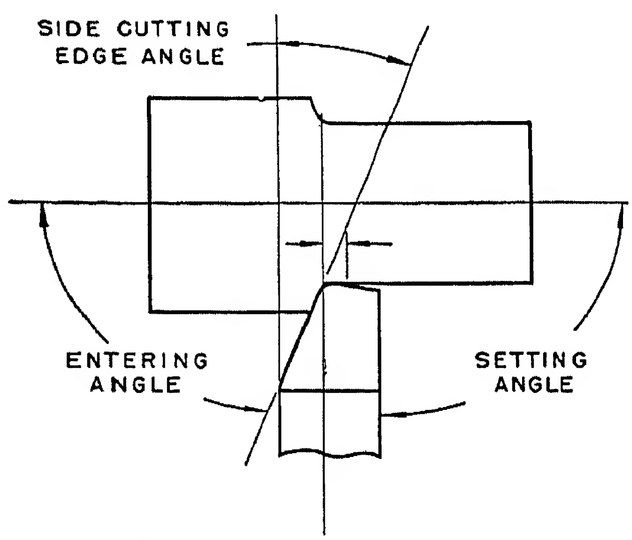


Fig. 4-8. The positioning angles of a tool.

The actual effect of the cutting edge angles in practice depends upon the angle to which the shank of the tool is set with respect to the workpiece, as indicated by Fig. 4-8. The entering angle is 180° minus the sum of the side cutting edge and setting angles.

The physical condition of the cutting edge and face of a tool has a considerable effect upon performance. A tool that is ground so that the cutting edge is jagged will break down quickly. Also a keen cutting edge and a good finish on the tool face are helpful in minimizing the formation of a built-up edge and promote good surface finish on the workpiece. A good practice is to stone a high speed steel tool after it is ground to sharpen the cutting edge and improve the finish of the face. Carbide tools are usually finish ground with very fine grit diamond wheels. A carbide tool for cutting steel may have its edge chamfered or "dubbed" 0.002 to 0.005 in. at 45° by means of a hand hone to remove weak irregularities along the edge.

Chip breakers. A continuous-type chip from a long cut is usually quite troublesome. Such chips become tangled around the workpiece, tool, and machine members and are dangerous to the operator because they are hot and sharp. Chip breakers are means

for breaking continuous chips into small pieces as they are formed to be able to dispose of them easily. Three common types of chip breakers are shown in Fig. 4-9. The gullet type is a groove, the stepped type an offset, and the mechanical type a block fastened on the face of the tool.

The width of land between the cutting edge and a chip breaker and the size of the chip breaker depend upon the feed intended for a tool. The greater the feed, the wider the land and the deeper the chipbreaker. A land width of 0.015 to 0.030 in. is common. For

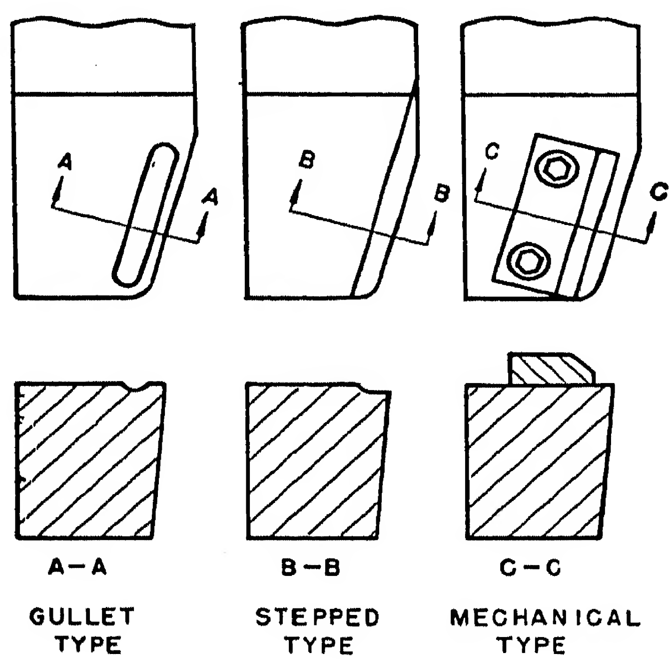


Fig. 4-9. Three common types of chip breakers.

a groove, a depth of 0.005 to 0.015 in. is recommended; and for a step, a depth of 0.015 to 0.030 in.

Tool shapes. Although the elements that have been described are found in all single point cutting tools, they are put together in many ways to satisfy various requirements. The most common shapes of tools used on lathes, shapers, planers, boring mills, etc. are illustrated in Fig. 4-10. These are only a few of the many modifications that are used.

Some side cutting tools are designed to cut in one direction, others in another. They are called right-hand or left-hand tools. The hand of a tool is revealed when the tool is placed with its nose pointing at an observer. If the cutting edge is on his right side, the tool is a right-hand tool, and vice versa.

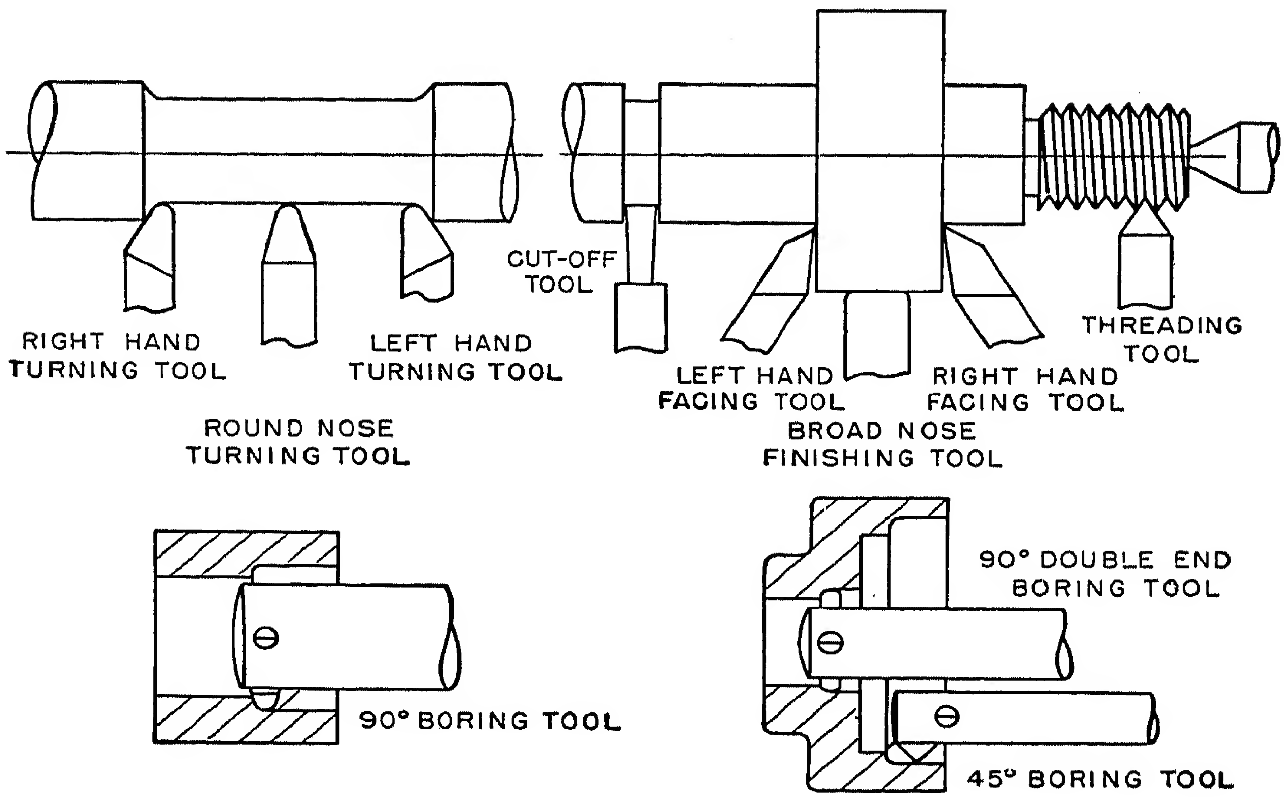


Fig. 4-10. Common tool shapes.

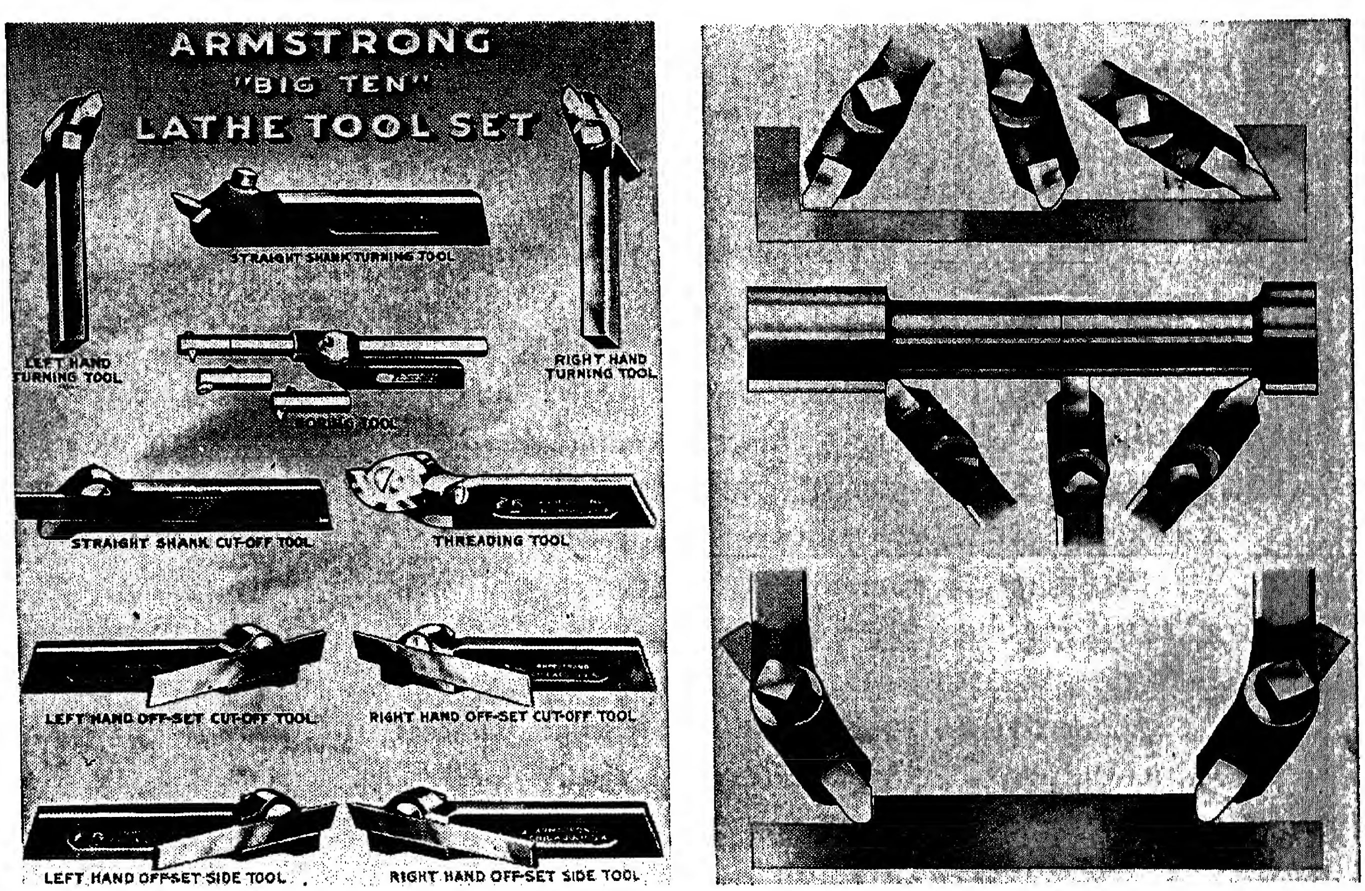


Fig. 4-11. Forged toolholders and bits and typical applications. (Courtesy Armstrong Bros. Tool Co.)

Single point tools. Single point cutting tools are formed in a number of ways to suit the various kinds of cutting materials. A tool may be formed by grinding the end of a piece of hardened tool steel as indicated by Fig. 4-5. That is also done to small pieces of high speed steel, but because of its cost that material is commonly welded on the end of a soft steel shank to make large tools. Sometimes steel tools are forged to shape before being finish ground.

Bits of all kinds of cutting materials are commonly clamped in toolholders. Forged tool holders and typical applications are shown in Fig. 4-11. A toolholder in a toolpost of a lathe is illustrated in Fig. 5-16. A straight, right-hand, or left-hand toolholder is chosen

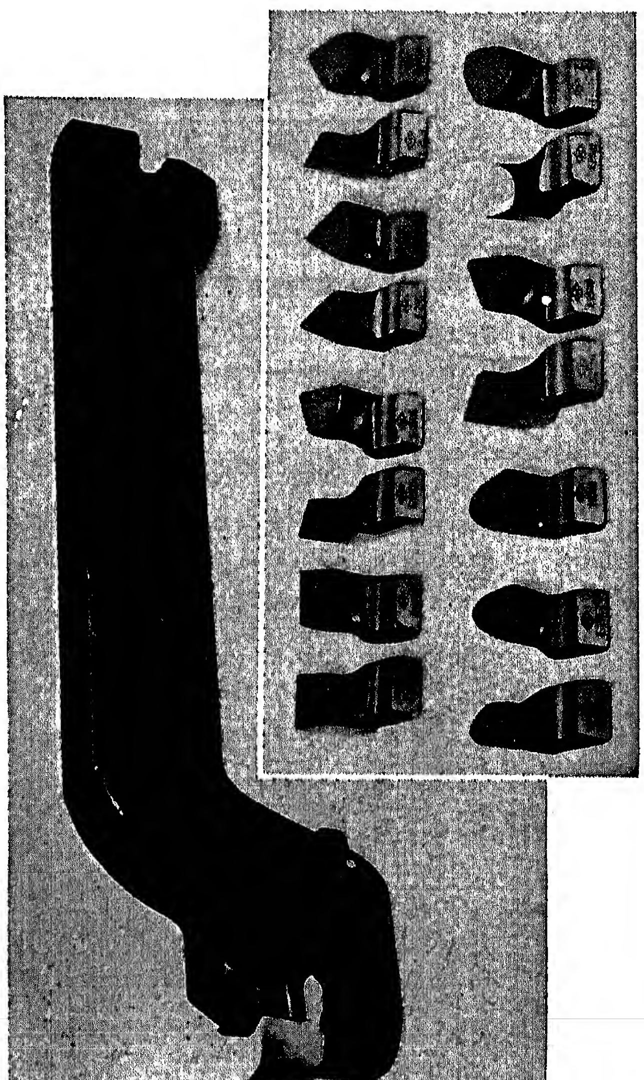


Fig. 4-12. A planer type tool bit holder with interchangeable bits. One end of the holder is goosenecked, the other is straight. (Courtesy The O.K. Tool Co.)

Fig. 4-13. Typical cemented carbide tipped tools. (Courtesy Kennametal, Inc.)

to suit each situation. Toolholders are made in various sizes for toolbits from 3/16 to 1½ in. square. The bit is inclined at an angle with the base in a toolholder intended for turning as shown in Fig. 4-11. Little or no grinding needs to be done on the top of a bit held in that way for the proper rake angle. Similar toolholders are also made in which the tool is not inclined. They are often used on shapers. Many other styles of toolholders are also available. One kind holds

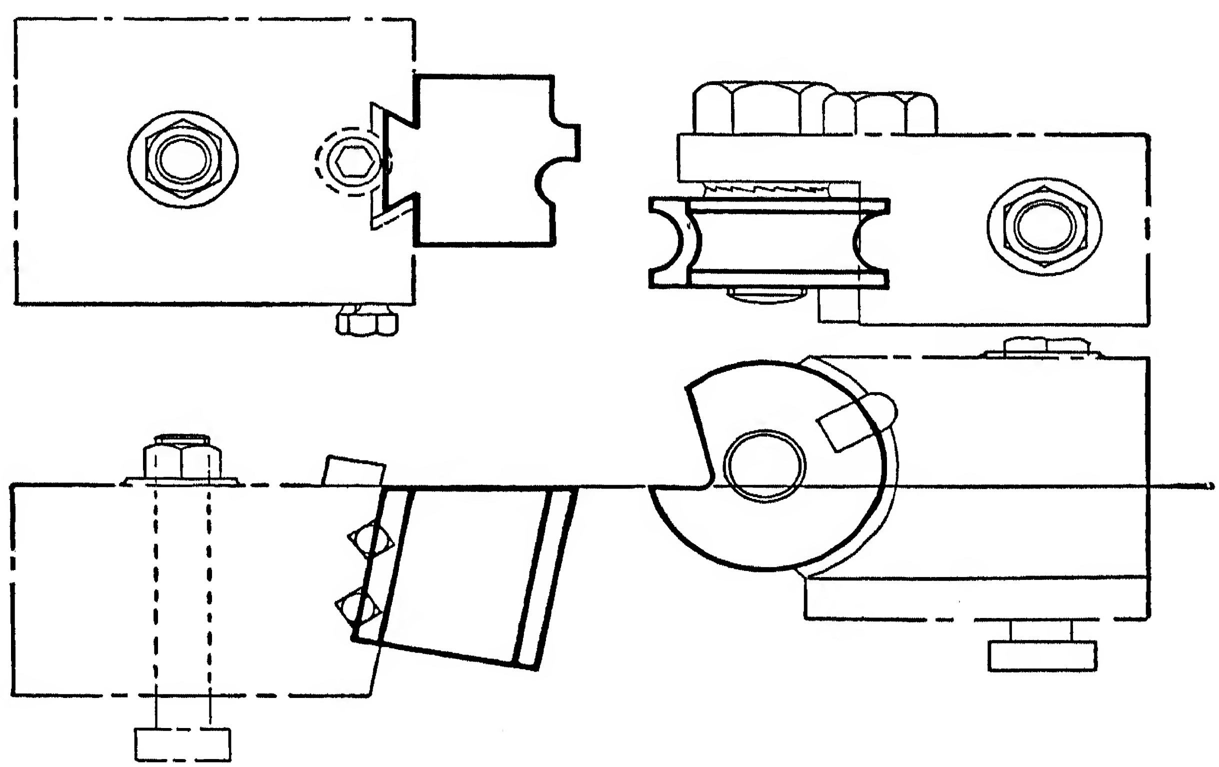


Fig. 4-14. Examples of flat and circular form tools and holders.

long narrow cut-off blades. Another kind is the spring type that has a flexible head and allows the bit to move away from the work if the cut is too heavy. It is used principally for threading tools. Holders are arranged to carry small and medium-size boring bars. Examples are shown in Figs. 6-9, 6-10, and 6-22. Special form tools of many kinds are carried in holders suited for each kind. A style of toolholder with bits of various shapes for planers is illustrated in Fig. 4-12. A broad nose finishing bit is clamped on the goosenecked end. Knurling rolls are mounted on special forged holders as shown in Fig. 6-29.

A tool bit like those of Fig. 4-11 should project from a toolholder a distance equal to about one and one half times its width, to allow sufficient chip clearance but prevent chatter. The toolholder should

extend beyond its support no more than enough to allow a wrench to be applied to the screw for clamping the tool bit.

Small pieces of expensive cutting materials are often brazed, welded, or clamped in slots or insets on the ends of heavy shanks of soft steel. Tools made in that way are said to be tipped. Cemented carbides are most commonly mounted in this manner. Typical carbide tipped tools are shown in Fig. 4-13.

Form tools. A form tool has a cutting edge with a definite profile or contour that produces a desired form on the workpiece. The action of a form tool is indicated in Fig. 1-1 E. A form tool may be made by grinding a profile on the end of an ordinary single point tool or bit. That is often done for simple profiles, like circular arcs, for lathe work. Two common types for production work are flat and circular form tools. Examples are given in Fig. 4-14.

A *flat form tool* has a square or rectangular cross section with the form along its side or end. A *circular form tool* is round and has a cut out segment that exposes the formed cutting edge. Form tools are sharpened by grinding the rake face behind the cutting edge. These form tools are used for the most part for producing pieces in large quantities on semiautomatic or fullyautomatic machine tools. Circular form tools are more expensive than flat form tools but last longer.

A variety of holders is used especially for form tools. Flat form tools are made with dovetails, tongues, grooves, and slots for locating and fastening them on their holders. A circular form tool has an axial hole, some plain and some threaded, for mounting on a screw or stud on its holder. A number of means are employed to keep the tools from rotating when cutting forces act on them. These include serrations on one side of the hub, locking pins, and keys.

The cutting edge of a circular form tool is usually below the center of the tool but is normally placed on the centerline of the work to give front clearance. A rake angle may also be provided. Flat form tools are ground and positioned at the desired angles and are fed radially into the work. A *skiving tool* is a type of flat form tool that is fed tangentially past the workpiece to produce a shaving action for finishing cuts. Because of the angles at which a form tool must be ground and positioned, it usually must be designed with a form somewhat different from that desired on the workpiece.

Multipoint tools. Cutters like twist drills, reamers, taps, and

milling cutters have two or more tool points each. They differ in overall appearance and purposes, but each cutting blade acts as and has the basic features of a single point tool. These kinds of cutters are usually identified with certain kinds of machine tools and will be described in more detail in connection with those machines.

Work Material

Some kinds of materials are easier to machine than others. In general, hard and tough materials are more harmful to tools, must be cut more slowly, and consume more power than soft materials. For instance, aluminum normally can be machined faster than steel. The ease with which a material can be cut is called its degree of machinability. A wide range of machinability may be found among varieties of any one class of material. For instance, steels with different alloying elements have different properties, but even a steel with a specific chemical composition may vary greatly in its machinability as it is heat treated in one way or another. The hardness and toughness of a steel do not alone determine how easily it can be machined. The size of the metallic grains and their structure are also important factors. That is also true of other metals. The grain size and structure of a metal, especially iron and steel, may be changed by heat treatment and cold working.

The properties of a metal desirable for long tool life and low power consumption are not always best for good surface finish. When most steels are raised to a red heat and cooled slowly, they are annealed. That makes them soft with a grain structure easy to cut, but a better finish is usually obtained if the metal is treated in a way that makes it harder.

The properties wanted in the material of a finished article often are not those desirable for good machinability. A finished part may be expected to have high strength or be quite hard. Common practice is to anneal or normalize steel parts in the rough, remove most of the stock by machining, heat treat again to get the desired properties for the finished product, and finish the surfaces by grinding when required.

Various elements are added to metals specifically to make them easy to machine. Such compositions are said to be *free machining*

or *free cutting*. They permit high cutting speeds with long tool life, well broken up chips, and smooth and accurate finishes. Moderate amounts of sulphur or lead are added to steel to make it free cutting. These additions form inclusions that weaken the structure of the metal to some extent and necessitate some sacrifice of strength in the finished parts. Nickel and molybdenum are added to cast iron, small amounts of selenium to stainless steel, lead to brass and bronze, and copper, lead, and bismuth to aluminum to improve machinability.

Cutting Speed

Calculation of cutting speed. The cutting speed of a tool is defined as the rate that it travels through the material in feet per minute (fpm), often referred to as surface feet per minute (sfpm). Suppose a shaft with an initial diameter of d inches is turned in a lathe. The surface of the shaft must travel past the tool at a desirable cutting speed of C sfpm. The lathe must be adjusted so that its spindle turns at N revolutions per minute (rpm) to drive the work at the proper speed in sfpm. In one revolution the work surface travels a distance of $\pi d/12$ feet past the tool. The required rate of rotation is then

$$N = \frac{C}{\pi d/12} = \frac{12C}{\pi d}$$

For practical purposes, π can be considered to be 3, and

$$N = \frac{4C}{d}$$

The highest surface speed on a revolving cutter, such as a drill or milling cutter, is found on its periphery. If C is the allowable cutting speed in sfpm of a revolving tool of diameter d inches, the speed at which it should be rotated is $N = 4C/d$ rpm.

Cutting speed and efficiency. Cutting forces and the power required to remove a cubic inch of stock decrease for many materials as the cutting speed increases. However, the decrease is only about 10 to 15 per cent over the range of speeds normally used.

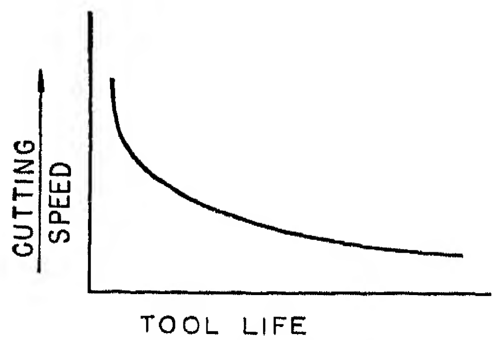


Fig. 4-15. A typical relationship between tool life and cutting speed.

Thus total power consumption varies almost in proportion to the cutting speed. A more significant effect of cutting speed is upon tool life.

As cutting speed increases, tool life decreases. If a test is run in which all other factors are kept constant, and the tool life T is measured for various cutting speeds, a relationship like that of Fig. 4-15 is found. Such a curve applies to only one set of conditions. If any of the constants is changed, another curve of similar form results, but in a different position. Such relationships appear as straight lines if plotted on log-log paper and are represented by the general equation that

$$VT^n = K$$

In this equation, V stands for the cutting speed in sfpm, and n and K depend upon the tool shape and material, work material, feed, depth of cut, and other cutting conditions. The exponent n varies in value as a rule from 1/5 to 1/10 and can be approximated in most cases by 1/6. The constant K equals the cutting speed for a tool life of one minute, and values for it are given in handbooks and other reference books.

The cutting speed-tool life curve does not in itself tell what cutting speed to use in a machining operation. Obviously, if a very low cutting speed is used, the tool will last a long time, but material is removed slowly, and the time consumed makes the cost high for doing the job. The cutting time and its cost can be reduced to a very small amount by running at a high speed, but the tool will have to be taken from the machine, sharpened, and reset often. Tool costs are excessive with too high a cutting speed. Somewhere between the extremes is a cutting speed that gives a minimum over-all cost. What that speed is for an operation depends not only on the cutting speed-tool life relationship but also on the relative costs of operating the machine and replacing and reconditioning the tool. In a plant where tool sharpening is systematized and a large volume of production is necessary to sustain heavy overhead

costs, relatively high cutting speeds are economical. On the other hand, in a jobbing shop where only a few pieces of any one kind are machined and tools are reconditioned one at a time, low cutting speeds are in order. An intricate cutting tool that is difficult to sharpen should be run at a low speed so that it does not have to be sharpened often.

Cutting speed is also important to surface finish. The tendency for a built-up edge to form when ductile materials are cut is diminished as the cutting speed is increased. Surface finish improves with the recession of the built-up edge. For that reason, finish cutting is usually done at higher speeds than rough cutting.

Many tables of cutting speeds are available. They differ in their recommendations because each is based upon a different set of conditions. Most published cutting speeds are given for production purposes. As a guide, the following speeds are those used in the Machine Tool Laboratory of the University of Illinois:

Table II

Average Cutting Speeds for Roughing Cuts with High Speed Steel Tools.

Material	*Cut speed sfpm*
Cast iron or cast steel	50–60
Malleable iron	70
Machine steel (bar stock)	60–90
Tool steel	35–40
Medium alloy steel	30–50
Yellow brass	150–200
Bronzes	30–80
Aluminum	150–300

For finishing cuts, speeds may be increased 25 to 50%. With cemented carbide tools, use speeds 2 to 3 times as high as shown in the table for high speed steel tools.

Feed and Depth of Cut

Definitions. *The feed of a cutting tool is the distance the tool advances into or along the workpiece each time the tool point passes a certain position in its travel over the surface.* In the case of turning on a lathe, the feed is the distance that the tool advances in one revolution of the workpiece. On a shaper, the feed is the distance the work is moved relative to the tool for each stroke. For single point tools, feed is specified in inches per revolution, inches per stroke, etc.

The depth of cut is the normal distance from the surface being removed to the surface exposed by a cutting tool. It is the thickness of stock taken off one side of a workpiece by a cut. The reduction in size of a diameter is twice the depth of cut taken to turn it. Depth of cut is measured in inches.

Feed and depth of cut and efficiency. The product of the feed in inches times the depth of cut in inches is the cross-sectional area of a cut in square inches. That area times the cutting speed in inches per minute gives the rate of stock removal in cubic inches per minute.

Cutting forces and power consumption increase as the cross-sectional area of a cut is increased with other factors constant. The increase is not proportional. The work required to remove a cubic inch of material is less for a heavy cut than a light cut, and from that standpoint heavy cuts are economical.

As the feed or depth of cut is increased, tool life is shortened. The cutting speed must then be reduced to get the same tool life, but when that is done, the amount of metal removed during the life of the tool is considerably increased. That means that a deep cut and a heavy feed with a low cutting speed are desirable to remove the most metal during a given tool life.

Although a maximum amount of metal can be removed during the life of a tool by large feeds and deep cuts, other factors often limit feeds and depths of cut in practice. The amount of stock available on a workpiece is one limitation. Heavy cuts require large amounts of power, and the power available from a machine tool may limit the size of cut. Heavy cuts produce large forces. The force that a cutter can stand and the deflections of machine and workpiece that impair accuracy may limit the size of cut. Heavy cuts tend to increase chatter, and that must be avoided. The finish required may dictate a light cut.

A heavy cut promotes the formation of a built-up edge and gives a poor finish. A large feed is much more detrimental to surface finish than a deep cut. If a large amount of stock must be removed in one cut, the best compromise for finish is to take deep cuts with light feeds. However, the best finishes can be obtained only from light finishing cuts and relatively high speeds.

The ideal feed and depth of cut for a machining operation depend upon circumstances. Large feeds can be used for soft work ma-

terials, smaller feeds for hard and tough materials. In many cases, cemented carbide tools will stand up under heavier feeds than other tool materials. A heavy feed and deep cut can be taken on a strong and powerful machine, whereas a light machine would not take the load. If a workpiece is frail and deflects easily, it can only be subjected to a light cut. A small tool cannot be fed as fast nor take as heavy a cut as a large tool. Some tools, such as boring tools for small holes, must be slight and cannot be made to take heavy cuts.

Because of many variables, no values of feeds and depths of cut can be specified to apply to all cases. The following general practices have been found economical for single point tools and are given as guides for average conditions:

The depth of cut for finishing is generally 0.010 to 0.030 in., but for roughing may be as much as ⅝ in. and even more. Feeds from 0.005 to 0.015 in. are generally used for finishing, but they may be larger with broad nose tools. Feeds for roughing are usually as coarse as the workpiece, tool, and machine will stand. When the depth of cut does not exceed ¼ in., a feed of 0.015 to 0.040 in. may be selected. For greater depths, a feed of 0.010 to 0.020 in. may be used.

Cutting Fluids

Purposes. The chief purpose of a cutting fluid is to cool the tool and workpiece, and the name of *coolant* is often given to the fluid. A tool lasts longer when heat is carried away fast enough to keep down its operating temperature. If a workpiece is overheated when cut, it may warp. A cutting fluid also helps lubricate cutting tools. In doing so, the fluid reduces friction forces and decreases power consumption. Lubrication decreases the abrasive action on the surfaces of a tool, reduces wear, and adds to tool life. The tendency to form a built-up edge is decreased by lubrication, and surface finish is improved. In addition, a cutting fluid is useful in cooling and washing away the chips formed in an operation. A fluid also should be capable of lubricating exposed machine elements, prevent corrosion, and not be harmful to the operator.

Investigations have indicated that a cutting fluid seldom reaches the edge of a tool that is cutting. Heat is conducted into the bodies

of the tool and workpiece from the cutting zone and then absorbed by the fluid. Lubrication of the chip on the tool face is accomplished back of the cutting edge. The effectiveness of a cutting fluid is diminished by high speeds and heavy cuts.

Kinds of cutting fluids. Two general types of cutting fluids are those mixed with water and those based on "straight" or "neat" oils.

Water is a good cooling medium but has little lubricating value and hastens rust and corrosion. Salts such as soda ash and tri-sodium phosphate sometimes are added to water to help control rust. Oil emulsions or "soluble oils" are the most popular water mixtures. These consist of compounds of soaps and soluble oils mixed with a large proportion of water to form a milky cutting fluid. Water solutions and the soluble oil emulsions absorb heat readily and have low viscosity and some oiliness depending upon concentration. As a rule they are chosen where cooling is of primary concern and lubrication secondary and give good results for most metal machining at low cost.

Three classes of cutting oils not used as emulsions are (1) straight mineral oils, (2) fixed or fatty oils, and (3) oils that are mixed with other compounds.

Straight mineral oils lack some of the unique qualities that distinguish cutting fluids and are used little. They have a specific heat about half that of water and a low degree of adhesion or oiliness but are very stable and do not develop disagreeable odors. They range in viscosity from kerosene used on magnesium and aluminum to the light paraffins for free cutting brass.

Fatty oils are oils from which soap can be made. At one time they were almost the only oils used in cutting. One of them, lard oil, is considered the best cutting oil for difficult work like threading and tapping. The fatty oils have a high degree of adhesion or oiliness, a relatively high specific heat, and their fluidity changes slowly with temperature. However, the fatty oils are expensive, become rancid and disagreeable in odor, and have a tendency to become gummy or dry when heated by cutting. They are used mostly at the present time for compounding cutting oils in which the good qualities of mineral and fatty oils are combined.

Sulphur and, to a lesser degree, chlorine are mixed with both

mineral and fatty oils to make *cutting oil compounds* that have high antiweld properties and promote free machining. These compounds play an important part in modern metal cutting practice, being used extensively on automatic screw machines, gear cutting, broaching, and various high production operations. They are particularly advantageous for tough, stringy, and unusually soft materials and help in producing good finishes and close tolerances on metals difficult to machine.

Questions

1. Describe what takes place when metal is cut.
2. Name and describe three types of chips.
3. Specify four considerations that are important for judging cutting tool performance and discuss each.
4. What six factors affect metal cutting efficiency?
5. What are the constituents of the common cutting tool materials? What are the advantages and disadvantages of each material?
6. Sketch a typical single point lathe tool and designate its angles.
7. What determines the amount of relief a tool should have? How does relief affect cutting tool performance?
8. How does rake affect the performance of a cutting tool? What conditions may determine the amount of rake a tool should have?
9. What effects do the cutting edge angles have upon cutting tool performance?
10. What is a chip breaker and what purpose does it serve?
11. How may the hand of a cutting tool be ascertained?
12. What are free cutting steels?
13. What is the ideal cutting speed to use in an operation?
14. Define feed and depth of cut.
15. What relative feed, depth of cut, and speed are desirable for rapid stock removal? Why?
16. Why are heavy cuts not conducive to good surface finishes?
17. What are the purposes of cutting fluids?
18. What are the principal kinds of cutting fluids? What are the advantages and disadvantages of each?

Problems

Please fill in the blank spaces below.

No.	Material	Diameter	RPM	SFPM
1	Cast iron	4″	______	______
2	C.I.	12″	______	______
3	C.I.	5′	______	______
4	C.I.	______	60	______
5	C.I.	______	230	______
6	Mach. steel	4″	______	______
7	" "	7″	______	______
8	" "	9″	______	______
9	" "	18″	______	______
10	" "	______	18	______
11	" "	______	35	______
12	______	9″	64	150
13	Aluminum	1″	______	______
14	"	5″	______	______
15	"	______	765	______
16	"	______	3060	______
17	Brass	4″	______	______
18	"	7″	______	______
19	Tool steel	¼″	______	______
20	" "	½″	______	______

References

American Society of Tool Engineers, *The Tool Engineers Handbook*, New York: McGraw-Hill Book Co., 1949.

Boston, Orlan William, *Metal Processing*, New York: John Wiley and Sons, Inc., 1951.

Carboloy Tool Manual GT-191, Detroit: Carboloy Co., Inc., 1949.

Cutting Fluids, New York: Tidewater Associated Oil Co., 1938.

Ernst, Hans, *Physics of Metal Cutting*, Cincinnati: The Cincinnati Milling Machine Co., 1938.

How to Run a Lathe, South Bend, Ind.: South Bend Lathe Works, 1938.

Kennametal Tool Manual, Latrobe, Pa.: Kennametal, Inc., 1947.

Kronenberg, Max, *Machining with Single Point Tools*, Cincinnati: The Cincinnati Milling Machine Co.

Machining of Metals, Cleveland: The National Refining Co.

Metal Cutting Lubrication in Theory and Practice, New York: Cities Service Oil Co., 1939.

Moltrecht, K. E. H., "Single Point Cutting Tools – Theory and Practice," *The Tool Engineer,* Vol. XXV, Nos. 5 and 6, Nov. and Dec., 1950.

Woodcock, Frederic L., *Design of Metal Cutting Tools,* New York: Mc Graw-Hill Book Co., 1948.

Manufacturers' Catalogs:

Armstrong Bros. Tool Co., Chicago, Ill.
Carboloy Co., Inc., Detroit, Mich.
The Cooper Bessemer Corp., Grove City, Pa.
Kennametal, Inc., Latrobe, Pa.
O. K. Tool Co., Shelton, Pa.
Super Tool Co., Detroit, Mich.
The Vascoloy-Ramet Corp., North Chicago, Ill.

Chapter 5

LATHES

LATHES ARE AMONG THE MOST BASIC, oldest, most versatile, and most widely used machine tools. They are commonly used for turning, facing, drilling, boring, reaming, and thread cutting. Other operations like milling, shaping, gear cutting, fluting, and grinding can also be done on lathes when more suitable equipment for such purposes is lacking.

Work is commonly rotated in a lathe either between centers or while held by a chuck, face plate, or fixture. A workpiece is supported and driven by the spindle in the headstock to the left of the operator's position. Between centers, the workpiece is supported at the outer end by the tailstock. These units are shown in Fig. 5-1. The headstock and tailstock are mounted on a bed. A carriage slides along the bed to feed the cutting tool. A cross-slide on the carriage provides infeed or cross-feed to the tool.

Principal Parts of Lathes

Headstock and spindle drive. The work spindle of a metal cutting lathe must be rigidly supported in heavy bearings that keep all radial and axial movements to a minimum. A lathe spindle is hollow to take long bar stock, and the hole is tapered at the front end to receive tapered center and tool shanks. Morse tapers are generally found on lathes. Tapered shank tools are knocked out by a bar or rod pushed from the rear through the hole in the spindle. The spindle nose in Fig. 5-2 is threaded to take a chuck, face plate, or dog plate. The spindle noses of some lathes are different; some are tapered; work-holding devices are bolted to the ends of some spindles. A chuck or face plate is located from a short pilot diameter

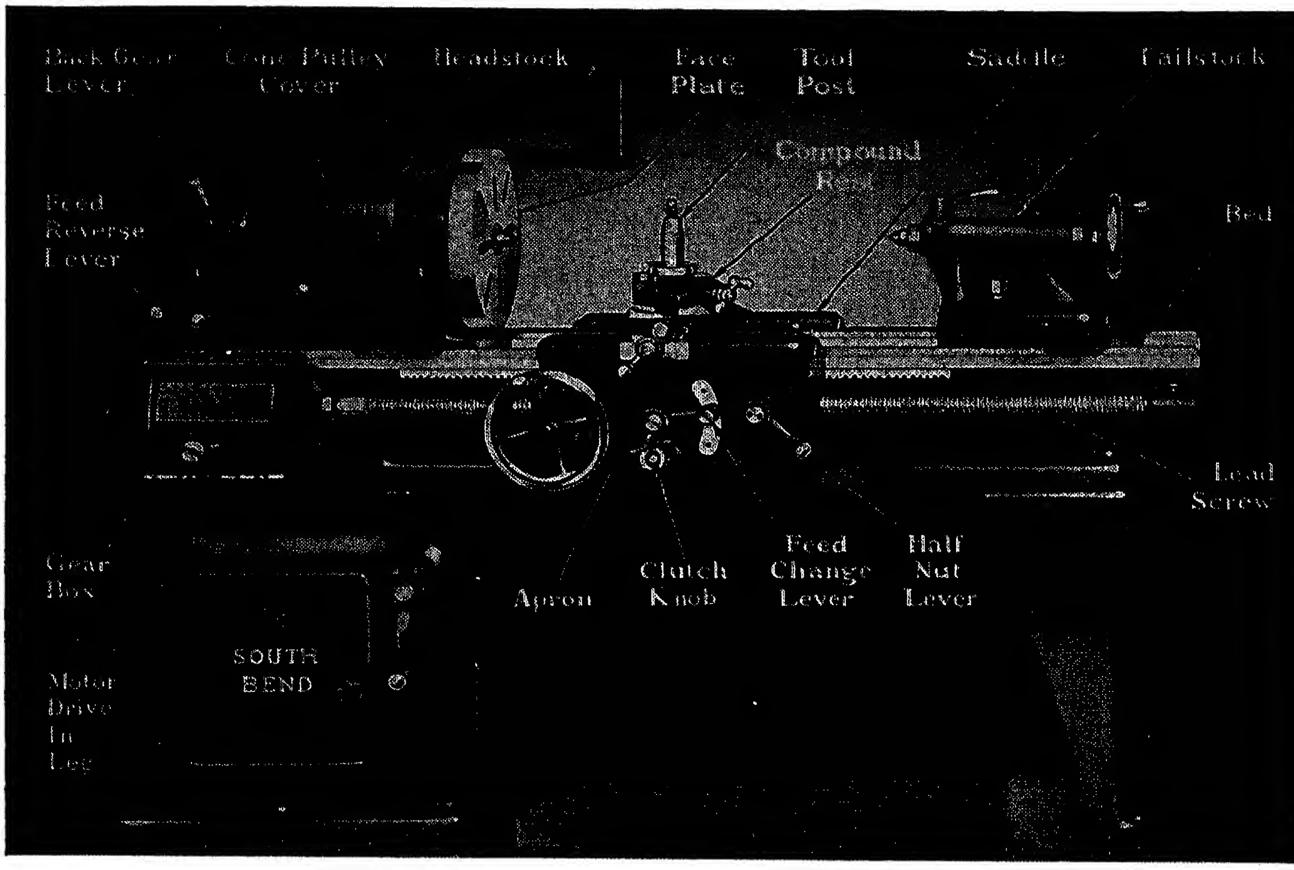


Fig. 5-1. A typical engine lathe. (Courtesy South Bend Lathe Works.)

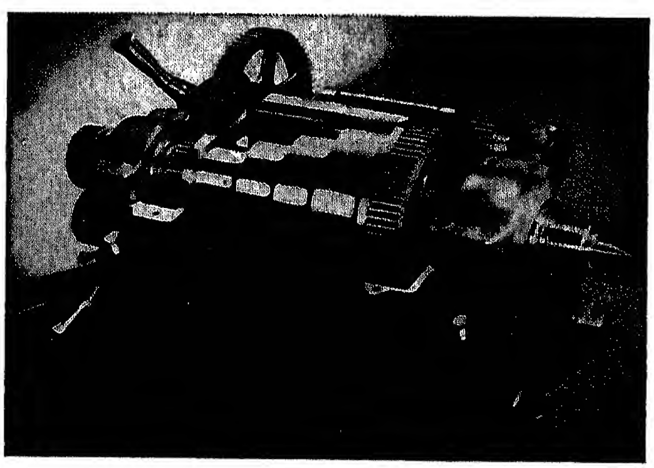


Fig. 5-2. A back geared belt driven headstock with gear guards removed. (Courtesy South Bend Lathe Works.)

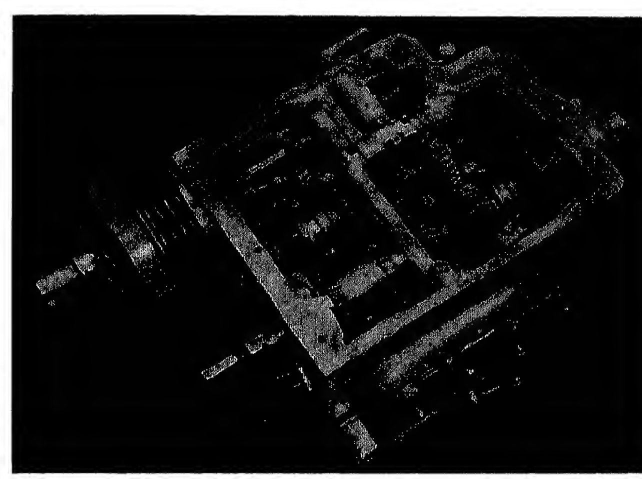


Fig. 5-3. An all geared headstock with top cover removed. (Courtesy The Monarch Machine Tool Co.)

and face on the spindle nose of the lathe of Fig. 5-4. Pins in the chuck or face plate slip into holes in the end of the spindle and are locked in place by quick-acting cams.

The spindle speed of a lathe can be changed to take care of different materials and workpiece sizes. The simplest drive has a step cone pulley and back gears as shown in Fig. 5-2. A large bull

gear is permanently fastened to the spindle just behind the front bearing. Next to the bull gear is a step cone pulley that is free to turn on the spindle. At the small end of the pulley is a spur gear. Behind the pulley are two back gears attached to a shaft that can be shifted toward or away from the pulley. When the back gears are brought forward, they mesh with the bull gear and the small gear on the pulley.

The cone pulley on the spindle is driven by another cone pulley (not shown in Fig. 5-2) on a countershaft. Present-day lathes, like other modern machine tools, have individual motor drives, usually of constant speed. The speed of the cone pulley is varied by changing the position of the belt on the steps. A series of four speeds is obtained with the back gears engaged. If the back gears are disengaged, the cone pulley can be locked to the bull gear to drive the spindle directly, and a higher series of four more speeds is available. When the back gears are engaged, the cone pulley and bull gear must be disconnected. Some lathes have double and triple back gears for larger speed ranges.

Many lathes, particularly large and powerful ones, have geared headstocks, typified by the one in Fig. 5-3. In headstocks of this kind, the gear combinations are changed, like in an automobile transmission, by shifting gears through levers on the outside or by hydraulic or electrical shifting mechanisms. Speed changes can be made quickly. Gears to give wide speed ranges and carry large amounts of power can be put into relatively small spaces. Speed ranges vary with the sizes and types of lathes.

Within a range, speeds are commonly varied in geometrical progression for lathes and many other machine tools. That means that each speed is multiplied by a constant to get the next one. For example, if the lowest speed is 12 rpm and the constant ratio is 1.5, the next speed is $12 \times 1.5 = 18$ rpm, the third speed is 27 rpm, and so on.

Tailstock. The tailstock may carry a center to support the outer end of a workpiece or may be used to hold a drill, reamer, etc. to machine work rotated by the headstock spindle. The tailstock spindle fits snugly in a hole in the body in line with the headstock spindle and parallel to the ways. The spindle can be moved a few inches in and out of the hole by turning a handwheel. The whole tailstock is slid along the ways on the bed for longer movements.

The tailstock spindle is generally graduated to show how much it is moved when feeding tools. The spindle can be clamped to keep it from moving, and the entire tailstock can also be clamped to the bed. The spindle is hollow with a tapered hole at the end to take the shanks of centers, drills, reamers, drill chucks, etc. Quite commonly when the spindle is retracted all the way, a pin ejects tools held in the taper. The body or top of the tailstock can be adjusted crosswise on its base either to align the tailstock for straight work or to offset it for turning tapers.

Bed. The bed of a lathe supports and maintains the alignment of the headstock, carriage, and tailstock. It must be massive and strong

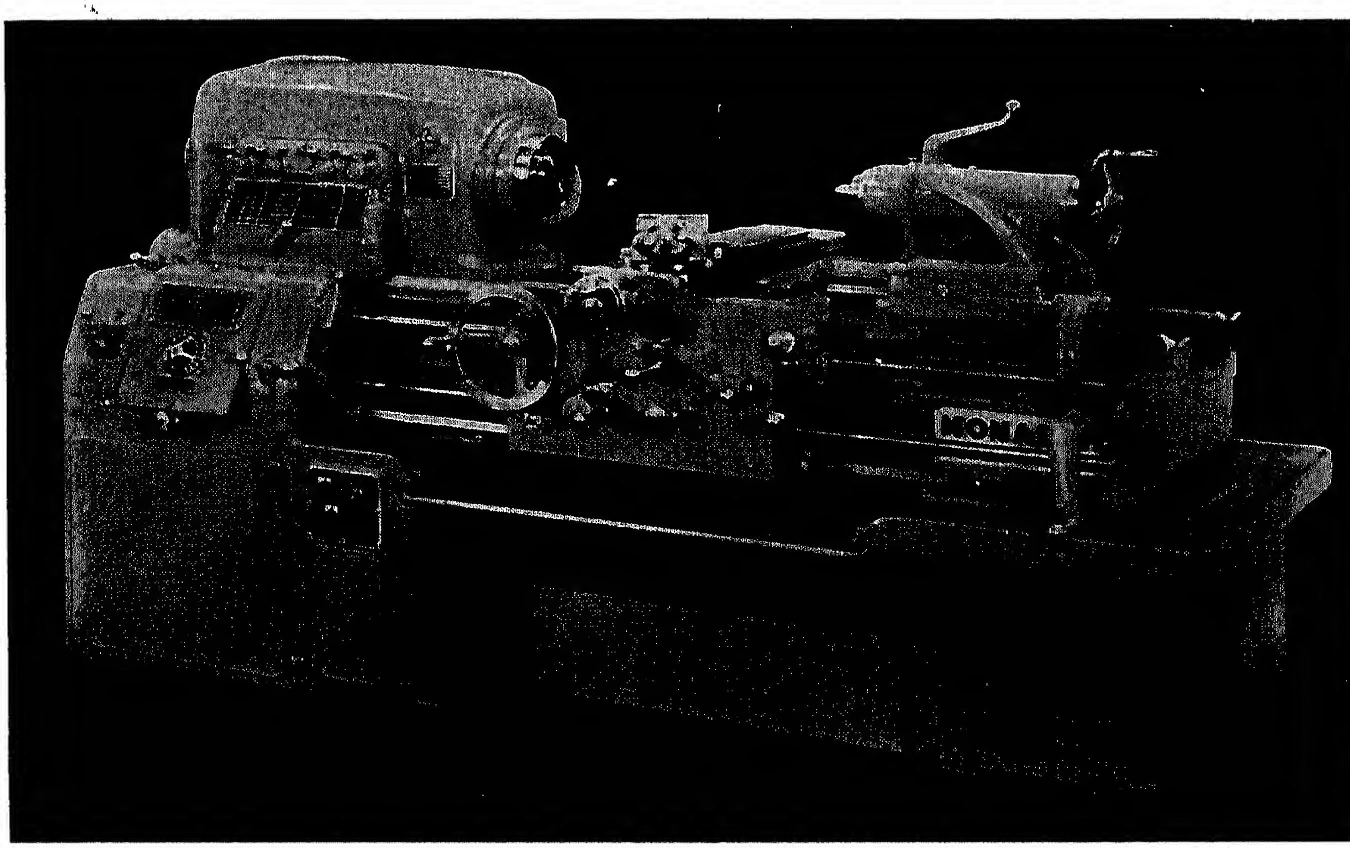


Fig. 5-4. A 20 in. completely geared engine lathe. (Courtesy The Monarch Machine Tool Co.)

to prevent appreciable deflection and twisting under the cutting forces and the weights of the other units and workpiece. The ways on top of the bed are scraped or ground to a high degree of truth so that the machine movements are precise. The ways are generally part of the bed and are of cast iron or cast steel which provide good bearing surfaces. Hardened steel ways are attached to the top of the bed on some lathes. Two sets of way surfaces are common on lathes: one set for the tailstock and the other for the carriage.

Carriage. The carriage of a lathe has several parts that serve to support, move, and control the cutting tool. The *saddle* is an H-shaped casting that bridges and slides along ways on the bed. On top of the saddle is a *cross-slide* that moves in a direction at right angles to the axis of the spindle. It is moved and adjusted by a screw and nut controlled by a crank and graduated dial on the front of the saddle. Each graduation on the dial of some lathes stands for a movement of the tool of 0.001 in., on others, for a reduction in diameter of 0.001 in.

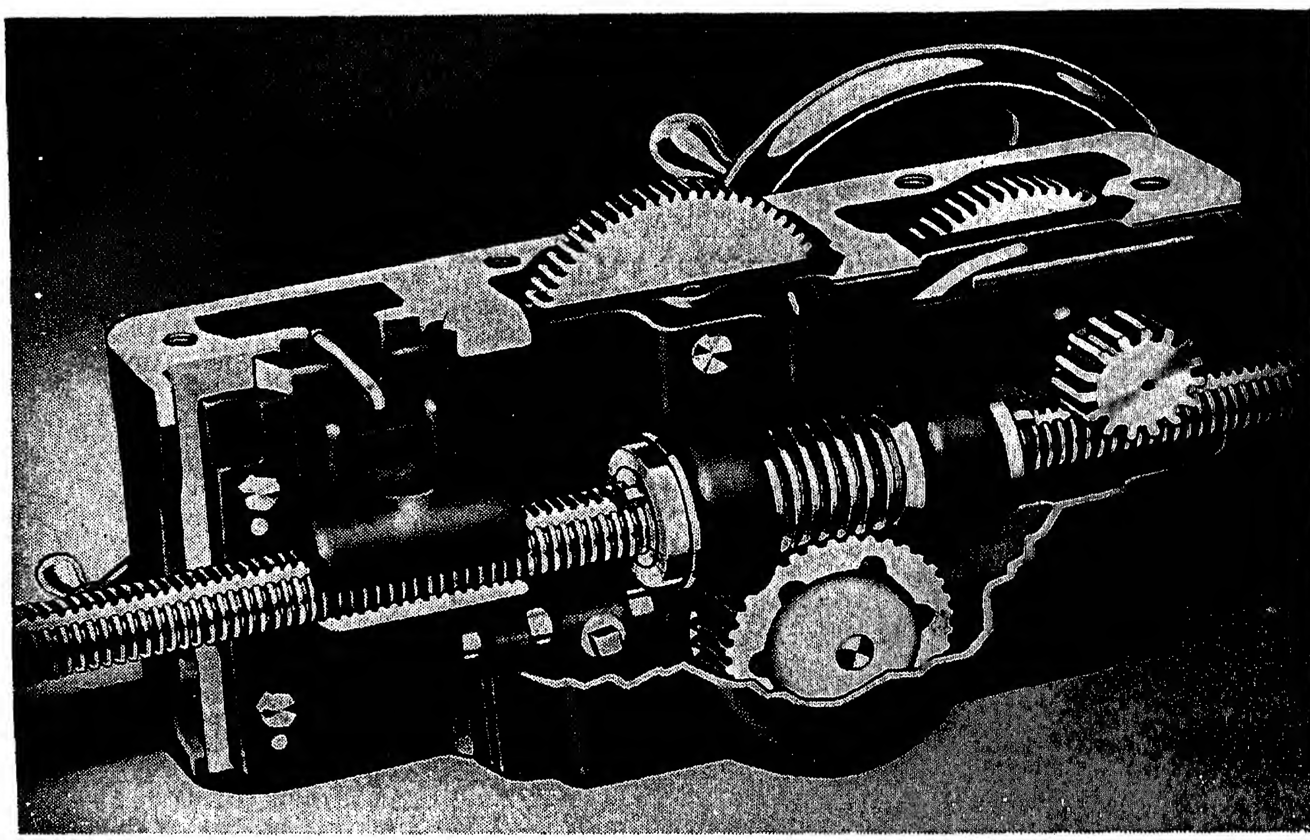


Fig. 5-5. A cut away view of the back of a lathe apron. (Courtesy South Bend Lathe Works.)

The commonest device for clamping the tool or toolholder to the lathe is a simple toolpost, shown in Fig. 5-1. The toolpost may be mounted directly on some cross-slides but usually is mounted on another unit called the *compound rest*. That unit can be swiveled to any position in a horizontal plane and clamped in the position selected. Its base is graduated in degrees to show the angle of swivel. The compound rest has a slide that can be moved a few inches in the direction to which the rest is swiveled. The compound rest slide can be positioned by a screw and nut with a crank and

graduated dial like the cross-slide. It is used to turn short steep tapers.

The bracket that hangs from the front of the saddle is the *apron*. It contains the mechanism for driving the carriage. A cutaway view of the rear of an apron is shown in Fig. 5-5. Turning the handwheel on the front of the apron turns the small pinion extending from the rear of the apron. The pinion engages a rack attached to the bed and pulls the carriage along.

Power comes to the apron through the leadscrew and feed rod. On many lathes these are separate shafts, but the leadscrew of Fig. 5-5 has a keyway and serves both purposes. The leadscrew passes through a split- or half-nut. When the half-nut lever, labeled in Fig. 5-1, is thrown, the half-nut is closed and engages the leadscrew. Then, as the leadscrew turns, the carriage is moved along at a definite rate for cutting a thread.

The feed rod is used to save the leadscrew when other than threading operations are done. A worm in the center of Fig. 5-5 is keyed to, turns with, and slides along the feed rod as the carriage moves. The worm meshes with a worm gear and drives a train of gears in the apron through a clutch engaged by the knob on the front of the apron. The power is applied to drive either the saddle along the bed or the cross-slide on the saddle by shifting the feed change lever on the front of the apron.

Feed change gears. The power feed drive on a lathe is always taken off the headstock spindle. In that way the feed, whether for turning or thread cutting, is always related to the speed of the spindle. In Fig. 5-6, the gear on the rear of the headstock spindle meshes with one of the reverse gears. The reverse gears are mounted on the reverse lever arm which may be shifted to bring either one or both reversing gears into engagement to drive the stud gear forward or backward. Change gears are mounted on and between the stud gear shaft and the leadscrew shaft to cut various pitches of screw threads and to provide desired rates of power feed. Thus, with a definite set of gears in place, the carriage travels a definite distance in inches for each revolution of the spindle no matter what the spindle speed. A lathe of the kind illustrated in Fig. 5-6 is known as a *standard change gear lathe.* It is simple and is used where feed changes need not be made often.

A *quick change gear lathe* does not have change gears that are

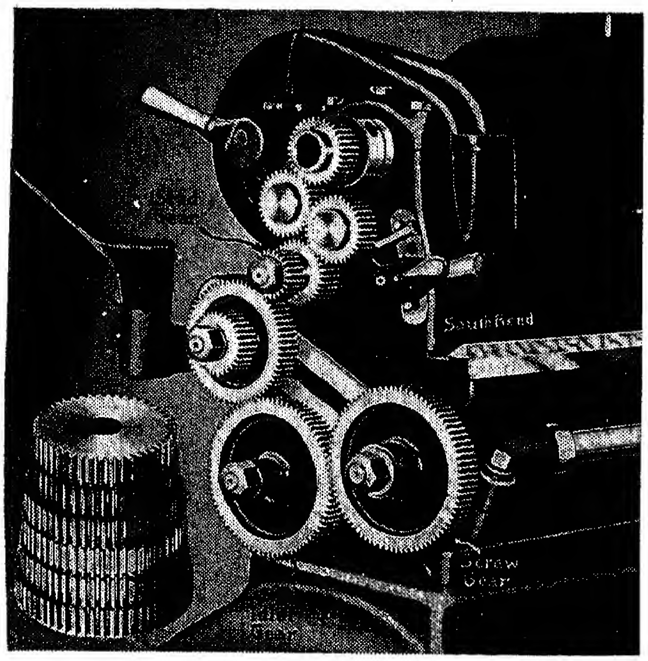


Fig. 5-6. Change gears on a lathe. (Courtesy South Bend Lathe Works.)

put on and taken off but instead has a quick change gear box on the left of the bed below the headstock, as shown in Figs. 5-1 and 5-4. The feed drive goes from the spindle to the change gear box in which gears are shifted by levers on the side to change the feed rate. Feed changes can be made quickly. A chart on the box tells the operator how to select each feed.

Most lathes have a large range of feeds. For example, a typical 14 in. general-purpose lathe cuts 66 different threads from 2 to 120 threads per in. and has 66 feeds from 0.014 to 0.084 in. per revolution. In general, larger lathes have fewer feed changes. Smaller lathes often cut more threads and have a larger feed range.

Sizes and Types of Lathes

The size of a lathe designates the largest nominal diameter in inches that can be swung over the ways and the over-all length of the bed in feet. Frequently, lathes are generously proportioned and will swing a workpiece of larger than nominal diameter over the ways, but the diameter that can be swung over the carriage is always less than the nominal diameter. The headstock and tailstock cover some of the bed, and therefore the length of workpiece that can be accommodated is less than the stated bed length. A typical 12 in. by 6 ft engine lathe is capable of swinging 14½ in. diameter over the ways but only 8 in. over the carriage. Its bed is 6 ft long but a piece only 30 in. long can be put between centers with the tailstock flush with the end of the bed.

More than just the size often needs to be considered in selecting a lathe. Complete specifications on lathes are given in manufacturers' catalogs. The motor horsepower is an indication of the rate at which metal can be removed. For instance, some 12 in. lathes

have 3 hp motors, and others 5 hp motors. The diameter of the hole in the spindle limits the size of stock that can be passed through it. Speed and feed ranges vary among lathes of the same rated size. The type of lathe indicates the class of work for which it is best suited. In any class of lathe, there are many variations in design, as is true of all machine tools. No description can be given in a reasonable space of all varieties of lathes, but the general types will be described in the paragraphs that follow to give a composite picture of present-day lathes.

Speed lathe. A speed lathe has only a few units: a headstock, tailstock, and toolpost mounted on a light bed. The headstock spindle rotates at high speeds, from 1200 to 3600 rpm, and usually only two or three speeds are available. Speed lathes are used for polishing, metal spinning, and woodturning. The tools are controlled by hand and commonly supported on a fixed T rest.

Bench lathe. Small lathes with beds up to 6 ft long and able to swing diameters up to 12 in. are commonly set on benches and are called bench lathes. In most cases they are as complete as larger lathes, only smaller in size and suitable for light work. Typical bench lathes have spindle speeds from 125 to 4000 rpm.

Engine lathe. The engine lathe is a common, general-purpose, and widely used type of lathe and is found in most plants. It received its name from being driven by an engine in the early days when many tools were driven by hand or animals. A power or engine lathe has mechanical means for feeding the cutting tools. Most engine lathes have all the units described as the principal parts of lathes. Some have extra units, like front and rear tool slides. Two typical engine lathes are illustrated in Figs. 5-1 and 5-4. Bed sizes generally range from 4 to 12 ft., and swing capacities from 9 to 50 in. A typical 12 in. engine lathe has 16 spindle speeds from 19 to 800 rpm. A large engine lathe with 50 in. swing has a speed range of 1½ to 65 rpm.

Toolroom lathe. A toolroom lathe looks like a regular engine lathe but is built more accurately, has a wide range of finely adjustable speeds, and is equipped with many accessories. A typical 10 in. toolroom lathe has a stepless spindle speed range from 4 to 2500 rpm. Such lathes are intended for precision work on tools, gages, dies, and exacting production parts. They possess a maximum

of convenience and versatility. Nominal sizes range from 9 to 30 in. for swing, but beds are shorter than on other lathes.

Duplicating lathe. Otherwise standard lathes may be equipped with attachments to enable them to turn, bore, and face all kinds of contours by duplication. Once a duplicating attachment is added to a lathe, it becomes a distinctive feature of the machine which is then called a duplicating lathe. On such lathes, the tool follows a path corresponding to the movement of a tracer finger guided along a template. Mechanical, pneumatic, hydraulic, and electrical devices are used to coordinate the movements of tool and tracer.

Fig. 5-7. A duplicating lathe equipped with an Air-Gage tracer. (Courtesy The Monarch Machine Tool Co.)

A typical job done on a duplicating lathe is shown in Fig. 5-7. A workpiece is turned between centers. As the tool is traversed, it is fed in or out by a hydraulic cylinder and piston attached to the tool slide. The tracer on the rear of the cross-slide is guided by the template on the rail bolted to the rear of the bed. The tracer activates a pneumatic circuit that governs the hydraulic infeed to the tool. Either flat or round templates may be used.

Production lathe. A production lathe is a simplified lathe intended for medium length production runs, generally for turning and facing operations. It has no leadscrew, few speeds and feeds, and often no compound rest. Tool posts are fastened to machined pads on the front and rear of the cross-slide.

Automatic lathe. Single spindle automatic lathes are used for simple operations to produce pieces 3/8 to 16 in. diameter and up to 84 in. long in large quantities. The machines are designed for speedy and easy operation. They are massive to resist heavy cutting forces and drive modern cutting tools to their utmost. Drives run as high as 75 hp. The cutting tools on these machines are carried on slides to cutting positions, cut automatically along predetermined paths to close tolerances, and then retract. All an operator has to do is

load and start the machine for each cycle. Often one operator runs two or more automatic lathes at the same time.

Automatic lathes are intended for reasonably long production runs d usually are not easy to change from one job to another. For in- ance, speeds and feeds are not changed by shifting levers, like on ngine lathes, but by removing and replacing change gears. To tear lown one job and set up another may take hours, as compared to a few minutes on an engine lathe. However, automatic lathes are fast when running and have been found economical for as few as six

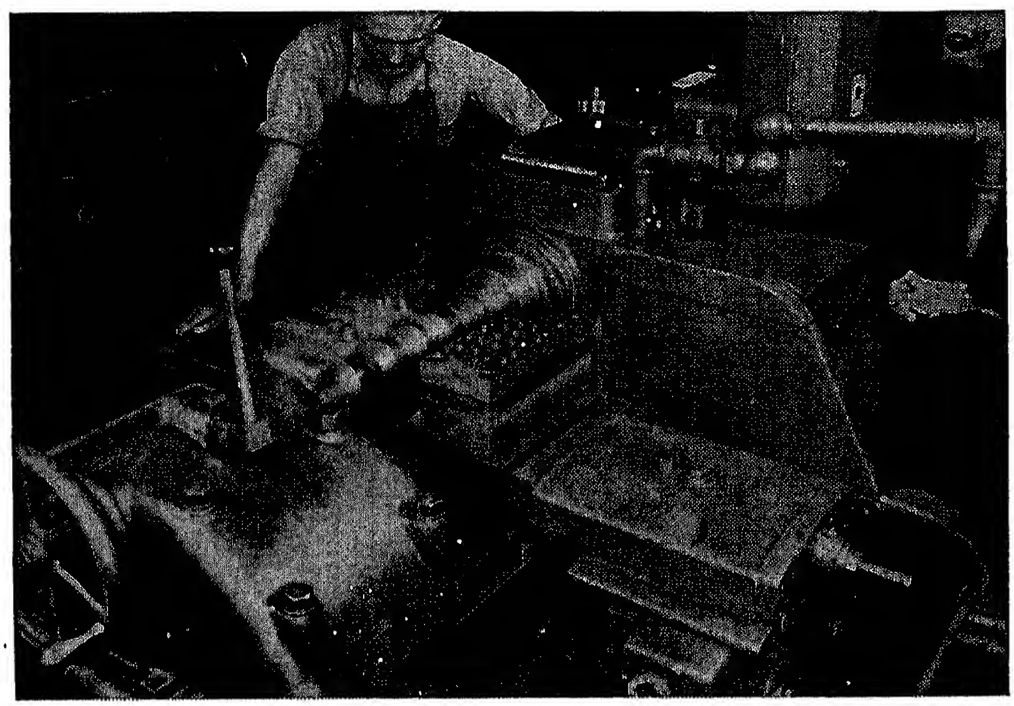


Fig. 5-8. An automatic lathe. (Courtesy Sundstrand Machine Tool Co.)

pieces in a lot. Careful scheduling to run similar jobs in succession saves change-over time.

A typical automatic lathe is shown in Fig. 5-8 with a job held in a chuck and supported by the tailstock. Chuck work without a tailstock can be done also. Quick acting air, electric, or oil operated chucks and special drivers and fixtures are often used for fast loading and unloading. Hydraulic or air operated tailstocks are common. The tools are mounted in blocks on the front and rear slides. The front tools are brought to the work, traversed over a preset distance, withdrawn, and taken back to starting position. When not cutting, the tools are traversed at a rapid rate. The paths of the tools may

be varied to cut multiple diameters and simple contours. Several tools may be carried on the front slide to cut multiple diameters and to machine long parts with short feed strokes. The rear tool slide normally feeds toward the center of the workpiece for facing, necking, grooving, forming, etc. and may carry one or more tools.

Special-purpose lathes. Some lathes have characteristics that enable them to do certain work. The *gap lathe* has a bed with a removable section under the spindle nose so that large work can be swung. The large diameter swung must be short in length because the gap space is limited. Facing and boring are most readily done on gap lathes.

Wheel lathes are made for finishing the journals and turning the treads of railroad car locomotive wheels mounted in sets. The wheels on an axle are driven by two opposed headstocks on one kind of wheel lathe. On another, the headstock can be opened to receive an axle between two wheels. These machines may have two, three, or four carriages for the tools.

Oil country lathes are used to make and maintain oil well drilling equipment and are like engine lathes of comparable sizes but have holes from 7 to 16 in. diameter through their spindles to accommodate large workpieces. Large chucks are permanently fixed to both ends of their spindles.

Accessories and Attachments

Lathe accessories are common work and tool holders and supports. They include chucks, collets, centers, drivers, rests, fixtures, and mandrels. Attachments are devices applied to facilitate specific kinds of operations on a lathe. In this class are stops, ball turning rests, thread chasing dials, and taper, milling, grinding, gear cutting, turret, cutter relieving, and crankpin turning attachments.

Chucks. The most important kinds of chucks are the three jaw universal, four jaw independent, combination, air or hydraulic, drill, and two jaw chucks. They are made in standard sizes designated by body diameters from 1½ to 36 in.

A *universal* or *scroll chuck* has three jaws engaged with a scroll plate as shown by the cutaway view of Fig. 5-9. When any one of the three pinions around the chuck body is turned by a wrench, the scroll plate revolves and the jaws move in unison. The inside or out-

side of round, hexagonal, or other regular trisectable sections can be gripped by this chuck. The top of each jaw is removable, is located by keys, and is held by screws. Each chuck usually has two sets of jaws, one for external and the other for internal holding. Hardened serrated jaws are used on rough workpieces, and soft jaws for finished surfaces. A hole in the chuck body allows work to extend back into the spindle hole.

One type of universal or self-centering chuck is mounted on an adapter that attaches to the spindle nose of a lathe. Adjustment to correct for runout is provided by four opposed fine threaded screws between the chuck body and the adapter. By means of the screws, the chuck body is displaced slightly to correct for the runout of the jaws. Once adjustment has been made, successive pieces are clamped by opening and closing the jaws. One manufacturer guarantees that adjustments can be made and held so that runout does not exceed 0.0005 in.

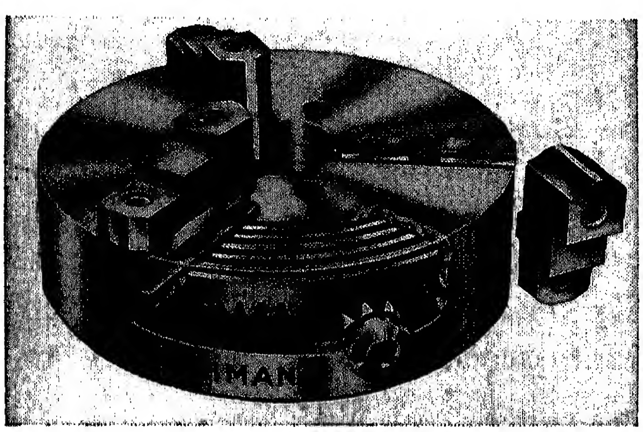


Fig. 5-9. A cut away view showing the mechanism of a three jaw universal self centering chuck. (Courtesy The Cushman Chuck Co.)

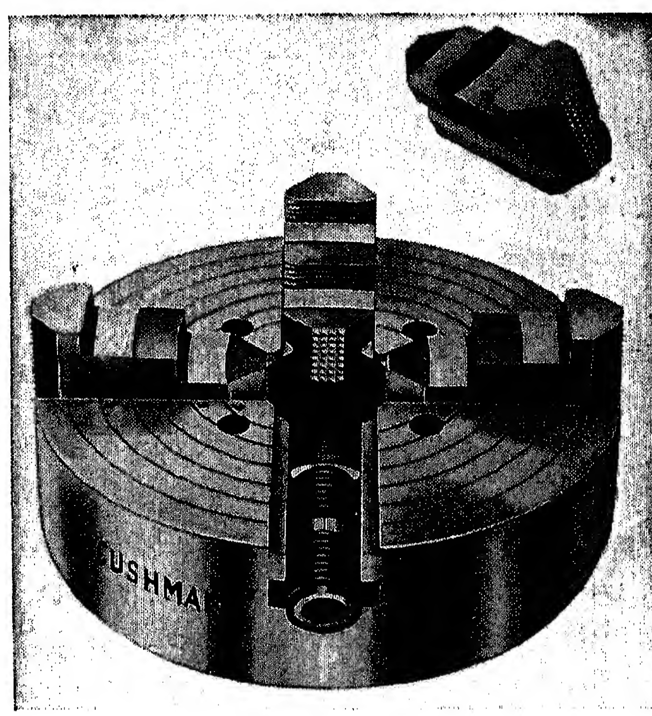


Fig. 5-10. A four jaw independent chuck. One jaw has been removed to show an operating screw. (Courtesy The Cushman Chuck Co.)

A *four jaw independent chuck* has jaws that are moved separately. A separate adjusting screw engages the teeth underneath each jaw as shown in Fig. 5-10. The jaws may be reversed for gripping either the inside or outside of a workpiece.

Four jaw independent chucks are suited for holding rough castings; square, octagonal, or irregular shaped pieces; workpieces that must have work done on them eccentric to the surfaces gripped; and

surfaces that must be made to run especially true. These chucks have more gripping capacity than three jaw chucks. The independent jaws can be adjusted to make a workpiece run within any degree of concentricity that is desired, but more time is needed to operate an independent chuck than a universal chuck.

A *combination chuck* combines the advantages of both the universal and independent chucks. It generally has four jaws that can be moved together through a scroll plate or adjusted separately by individual screws.

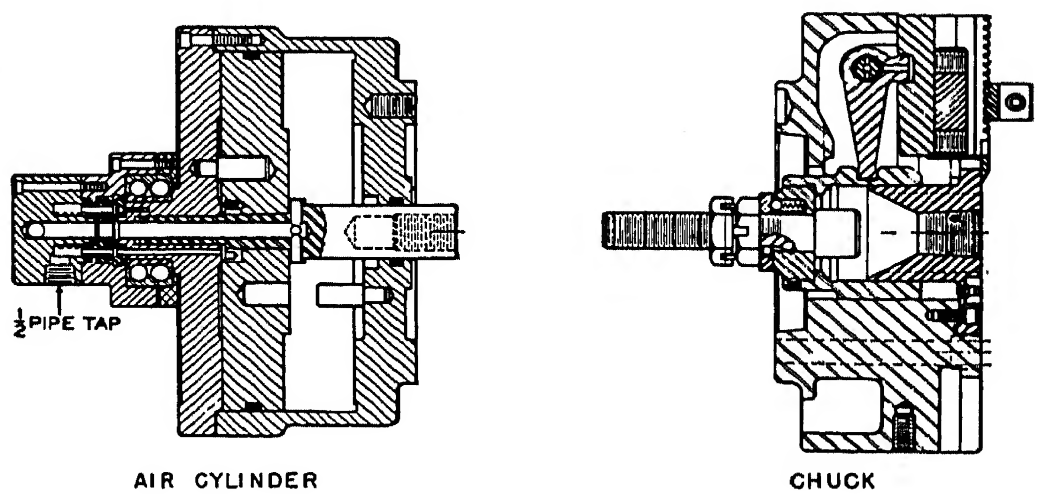


Fig. 5-11. A cross-section of an air cylinder and air operated chuck. (Courtesy The Cushman Chuck Co.)

Two jaw chucks are adapted to hold workpieces of irregular shapes by means of slip jaws added to the permanent jaws. That can be done for some pieces more readily on a two jaw chuck than on three and four jaw chucks. Sufficient production is necessary to justify the special jaws.

Air and hydraulic operated chucks are often used on lathes and other turning machines engaged in production. They are quick acting and grip the work strongly. An air cylinder and chuck are mounted on the automatic turret lathe in Fig. 21-9. A cross-section of such an air cylinder and chuck is shown in Fig. 5-11. The cylinder is mounted on the rear of and revolves wtih the machine spindle. The air pressure is controlled by a valve convenient to the operator. The force is transmitted from the piston to the chuck by a rod through the hole in the spindle and is distributed by a linkage to the jaws.

A *wrenchless chuck* is operated by a lever on a ring on the rear of the chuck body as illustrated in Fig. 5-12. The lever does not re-

volve with the chuck. Handling of a loose wrench is eliminated, clamping and unclamping are done by simple direct motions, and it is not necessary to wait for the chuck to come to rest to loosen and tighten it. Consequently, wrenchless chucks are fast in operation and often are used on production.

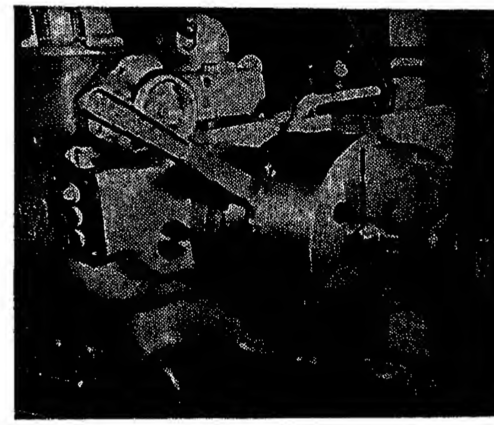


Fig. 5-12. An application of a wrenchless chuck. (Courtesy The Warner and Swasey Co.)

A *drill chuck* may be used on either the headstock or tailstock spindle of a lathe to hold straight shank drills, reamers, taps, or small diameter workpieces. The construction of one type of drill chuck is shown in Fig. 5-13. Tools are gripped by the jaws wedged by the inside taper of the shell. The shell is fastened to the body that is screwed up or down on the shank to close or open the chuck. That is the basic action of most drill chucks. The particular chuck shown can be opened or closed while revolving. The shell is gripped and pushed up to tighten the chuck, and the sleeve is held and pulled down to loosen the chuck. Most drill chucks can only be operated by turning their sleeves by hand or with a key while they are standing still. A drill chuck key is a small pinion with a pilot on one end and a handle on the other. The pilot is inserted in a hole in the side of a drill chuck, the pinion engages teeth around the chuck sleeve, and the handle is turned to tighten the chuck. Key-operated drill chucks are shown in Figs. 6-8 and 6-22.

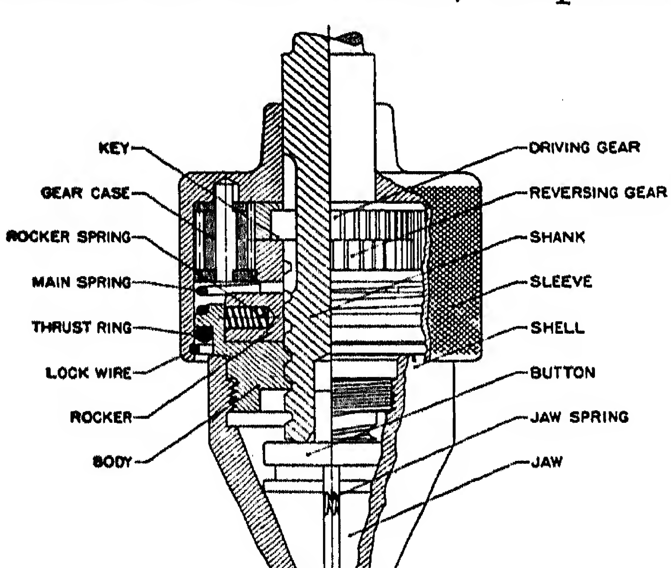


Fig. 5-13. A sectional view of a drill chuck. (Courtesy Wahlstrom Tool Div., American Machine and Foundry Co.)

Collets. A collet is a thin steel or brass bushing with lengthwise slots and an outside taper. When it is drawn or pushed into a tapered sleeve, its sides are sprung slightly together to grip a workpiece securely and accurately. Collets have round, square, hexagonal, and other kinds of holes for various shapes of stock. Collets are used for

bar stock up to about 2½ in. diameter and also for individual pieces. *Solid collets* are made in specific sizes, each for a definite workpiece size. To be accurate, a collet should not be sprung very far and should be used to grip pieces no more than 0.001 in. larger or smaller than its nominal size. Commercial collets have standard sizes. *Master collets* have large holes and are fitted with interchangeable inserts, also called pads, bushings, or jaws, for various sizes and shapes of workpieces.

The complete assembly utilizing a collet is called a *collet chuck*

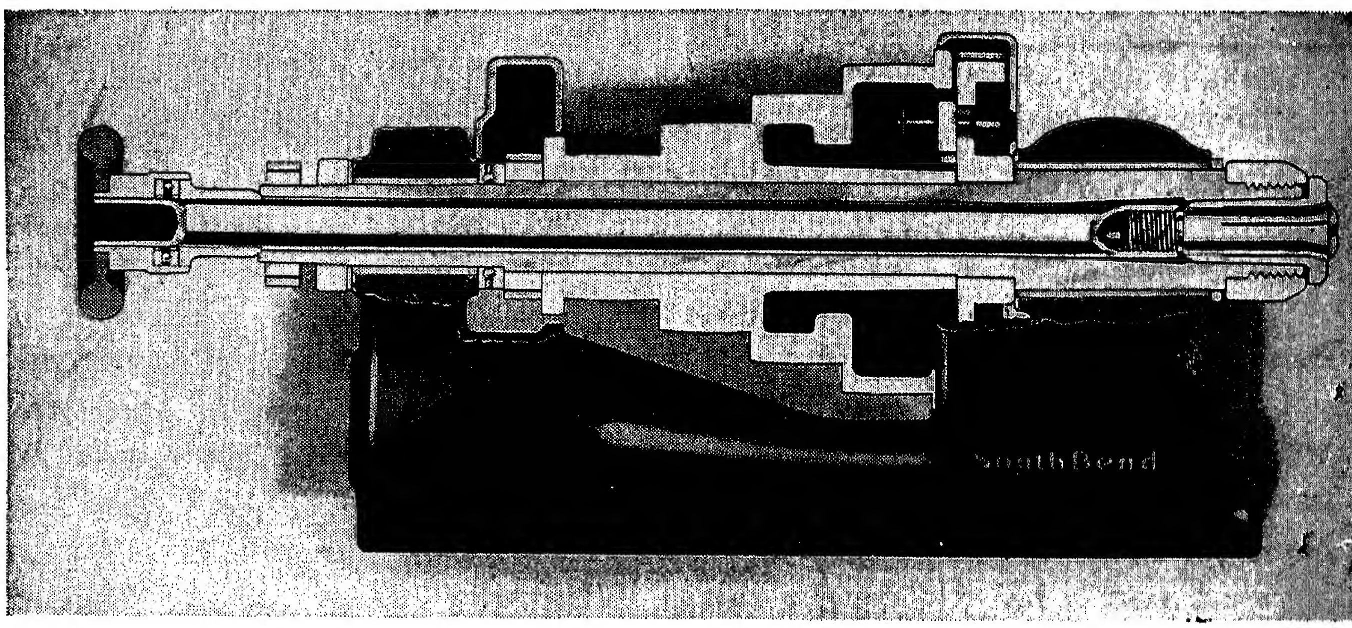


Fig. 5-14. A cross-section of a lathe headstock showing the construction of a draw-in collet chuck. (Courtesy South Bend Lathe Works.)

One of the simplest is shown in Fig. 5-14. A tube with a handwheel on its end runs through the hole in the spindle and is screwed on the threaded end of the collet. The handwheel is turned to draw the collet into the sleeve of the collet chuck. Instead of a handwheel, a lever and cam may be the means of drawing the collet. Another form of collet chuck has its handwheel around the chuck on the front of the spindle. Air or hydraulic cylinders are used to operate collet chucks for rapid production.

A *step chuck* works like a collet but takes larger diameters than ordinary collets and is used to hold disks, gear blanks, etc.

Centers and drivers. A lathe center has a tapered point with a 60° included angle to fit workpiece center holes with the same angle. The other end or shank of the center is tapered to fit the hole in the

headstock or tailstock spindle. Typical centers are shown in Fig. 5-15. The solid or dead center is the kind used most on lathes. A solid center mounted in the headstock spindle turns with the workpiece and does not need to be lubricated. A solid center in the tailstock spindle must be lubricated on its end because the workpiece turns on it. The ball bearing center of Fig. 5-15 is one kind of live center that always turns with the workpiece. A live center requires less attention but has a certain amount of looseness and is not as accurate as a dead center.

Most centers are made of tool steel, hardened and ground. For sever service, center tips of high speed steel, nonferrous alloys, or cemented carbides are attached to machine steel shanks.

A workpiece mounted between centers must be driven in a positive manner. This is commonly done by clamping a dog to one

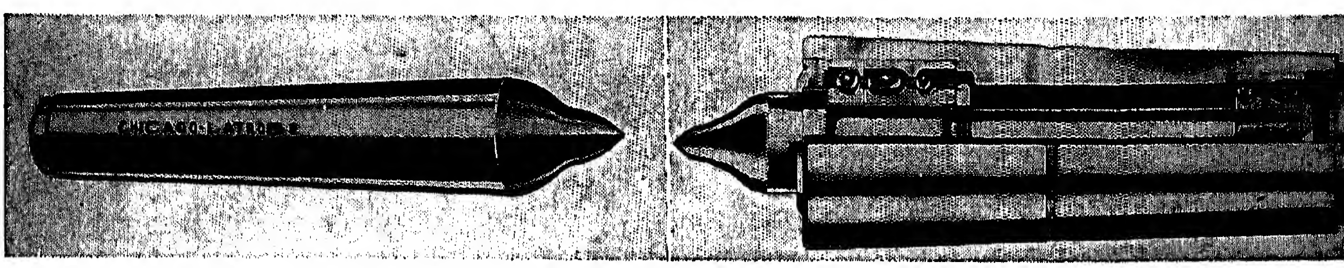


Fig. 5-15. (left) **A solid center. (Courtesy Chicago-Latrobe Twist Drill Works.)**
(right) **A ball bearing live center. (Courtesy Ready Tool Co.)**

end of the workpiece. A standard dog is clamped to a workpiece and engaged with a dog plate in Fig. 5-16. A piece of brass or copper may be placed under the clamping screw to keep it from defacing a finished surface. Another form of dog is shown in Fig. 5-19. Slots in the dog plate take the tail of the dog. One of the slots is open to distinguish it from the others and provide a means of identifying the dog's position. That is helpful in thread cutting where the workpiece must always be put back in the same position if it is taken from between centers. Dogs are made in a variety of sizes and shapes.

Automatic drivers often take the places of dogs and dog plates for quick loading in production. In general, an automatic driver resembles a chuck, but its jaws swing. The workpiece is placed between centers. When the driver turns, each jaw swings against the work and makes contact on a cam-shaped surface that is tightened against the workpiece by the driving torque. The time

for removing, placing, and tightening a dog on each workpiece is saved.

Face plates and fixtures. A face plate is mounted on the lathe of Fig. 5-1. It is larger in diameter than a dog plate and has a number of radial slots for bolts. Workpieces too large to chuck conveniently or irregular in shape, like castings, are often bolted to the front of a face plate. If the workpiece needs to be located in a certain position, it may be indicated and shifted around until true and then clamped firmly.

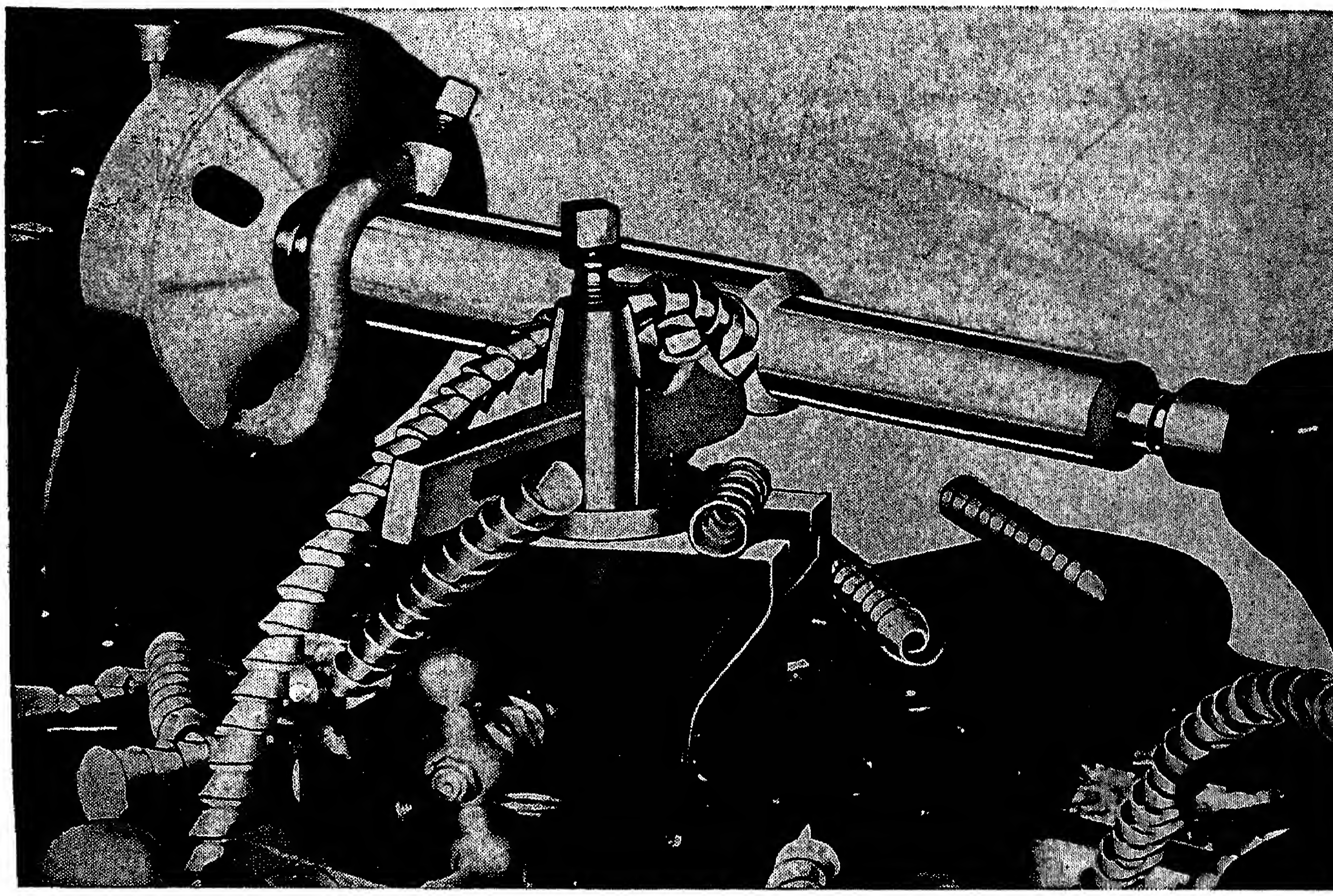


Fig. 5-16. Turning a steel shaft between centers. (Courtesy South Bend Lathe Works.)

A fixture is a special device for holding and locating a workpiece. Fixtures are commonly used for quantity production of pieces. They are more common on turret and automatic lathes than on engine lathes. A fixture may be fastened directly to the spindle nose or bolted on a face plate.

Mandrels. A mandrel locates a workpiece from a hole. A *solid mandrel* shown in Fig. 5-17 is the most common. It has a ground diameter, usually with a taper of 0.006 in. per ft, and a length

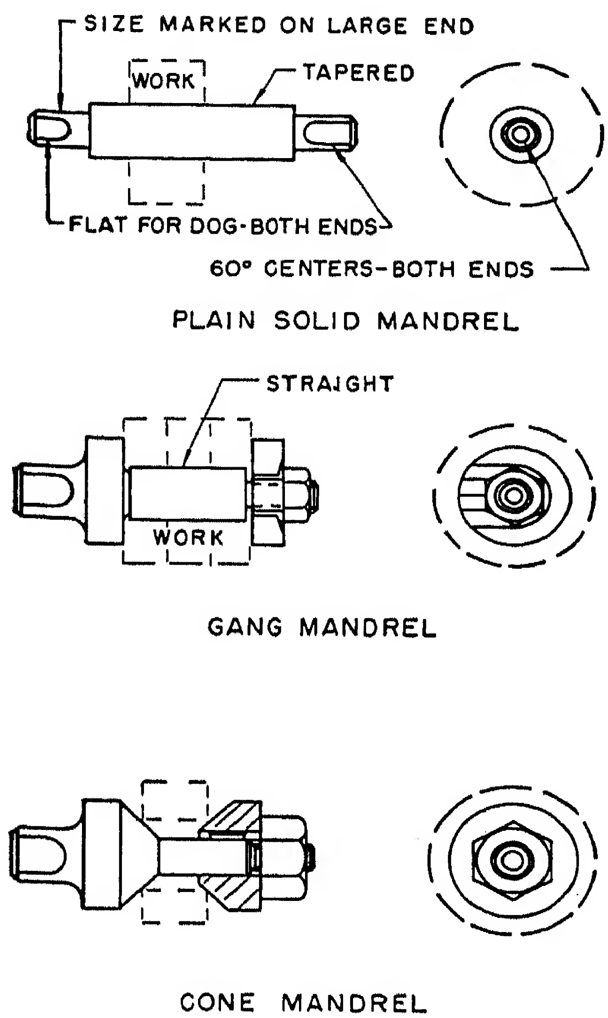


Fig. 5-17. Common mandrels.

from about 4 to 12 in. on which workpieces are pressed. It is well to lubricate a tapered mandrel so that the workpiece does not freeze on it. Commercial mandrels are available in standard sizes, but special mandrels are often made for in-between sizes. The hole in the workpiece must have a diameter between the small and large diameters of the mandrel, a matter of a few thousandths of an inch.

A mandrel that has a straight diameter must have a shoulder against which the work is held by a nut and washer. The *gang mandrel* illustrated in Fig. 5-17 has a nut smaller than the hole in the workpiece. A slotted C washer may be slipped off when the nut is loosened only a little, and the workpieces taken off and put on quickly.

A *cone mandrel,* shown in Fig. 5-17, can center pieces having a hole that varies considerably in size. Another type of mandrel has a split bushing that is expanded on a taper on the mandrel to fit the hole in the workpiece. That is called an *expanding mandrel.*

A mandrel is commonly mounted on centers and driven by a dog. Flats are provided on the ends for the dog screw. Some mandrels are made so that one end may be gripped in a chuck or have a tapered shank to fit the hole in the spindle.

Rests. Workpieces often need extra support. This is especially true of long thin pieces that are likely to spring away from the cutting tool. Two common means of support are shown in Fig. 5-18. A *center* or *steady* rest is clamped to the bed and has three adjustable shoes or jaws that contact the workpiece at equal intervals around the periphery. The center rest in Fig. 5-18 is between the carriage and the headstock. A center rest may also be used instead of the

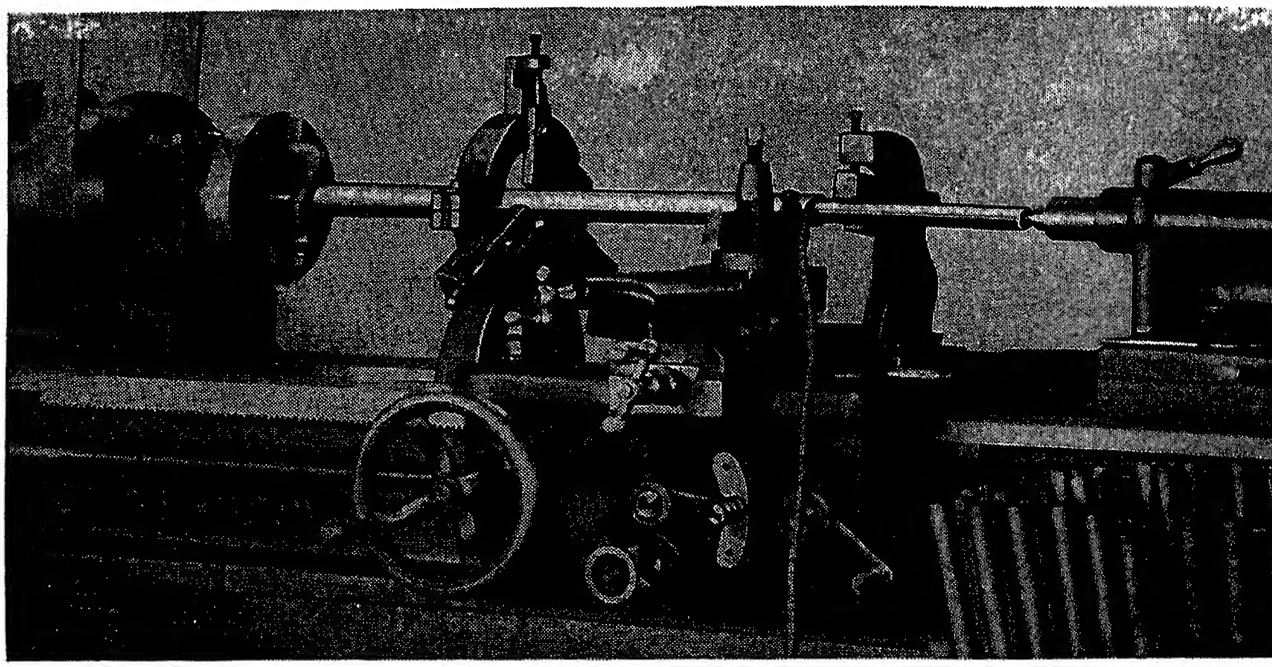


Fig. 5-18. A long slender shaft supported by both a center rest and a follower rest while being turned. (Courtesy South Bend Lathe Works.)

tailstock to carry the overhanging end of a workpiece that has its other end in a chuck. The overhanging end is then left open for drilling, boring, etc. A *follower rest* is fastened to and moves along with the carriage to support the workpiece opposite the cutting tool position.

Attachments. A *taper attachment* guides a lathe tool to cut a taper. A setup using a taper attachment is illustrated in Fig. 5-19. The attachment and its operation are described under "Taper Turning and Boring" in Chapter 6.

A *thread dial indicator* is useful in cutting threads, particularly long threads, because it shows the operator when to engage the half-nut to pick up the thread at the beginning of each pass to avoid damaging the thread. One is mounted on the right end of the apron in Fig. 5-4. The dial is on a shaft with a gear engaged with the leadscrew. The dial turns when the leadscrew is revolving and the half-nut is not engaged. Then if the half-nut is closed and the carriage moves along, the dial stands still. The dial has evenly spaced graduations, every other one numbered. As the dial turns, the graduations pass a fixed reference line. The half-nut is closed for all even-numbered threads when any line on the dial coincides with the reference line. For an odd number of threads per inch, the half-nut is engaged at any numbered line on the dial. The half-nut is engaged at any odd-numbered line on

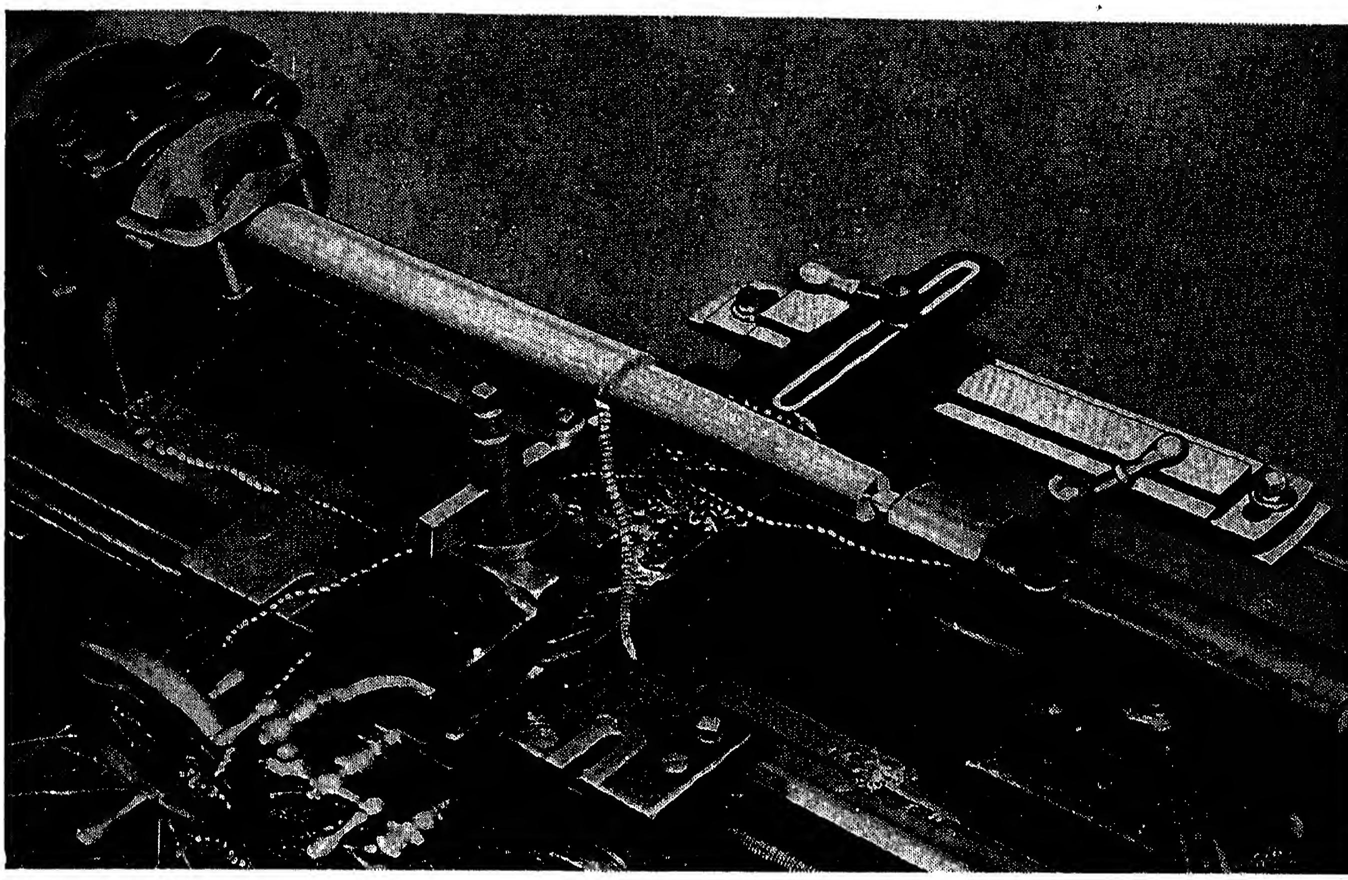


Fig. 5-19. Taper turning with a taper attachment. (Courtesy South Bend Lathe Works.)

the dial for half threads, like 11½ threads per inch. The carriage must be returned to the starting position for quarter threads.

A *milling attachment* consists of a vise that is carried on an angle bracket mounted in place of the compound rest on a lathe. A typical setup is shown in Fig. 5-20. The vise holds the work and can be swiveled in a vertical plane and a horizontal plane. The vise is positioned vertically by a screw and dial. Horizontal movements are obtained from the saddle and cross-slide. The milling cutter is rotated in the headstock spindle.

A large variety of work can be done with the milling attachment on a lathe, including the milling of T slots, keyways, dovetails, and small plane surfaces. The attachment is used mostly for odd jobs where a regular milling machine is not available.

A *gear cutting attachment* is carried on the cross-slide in place of the compound rest, like the milling attachment, and is arranged to hold and index a gear blank on an arbor. Teeth are cut in the blank by a gear cutter, usually held on an arbor between centers and driven by the headstock spindle through a dog.

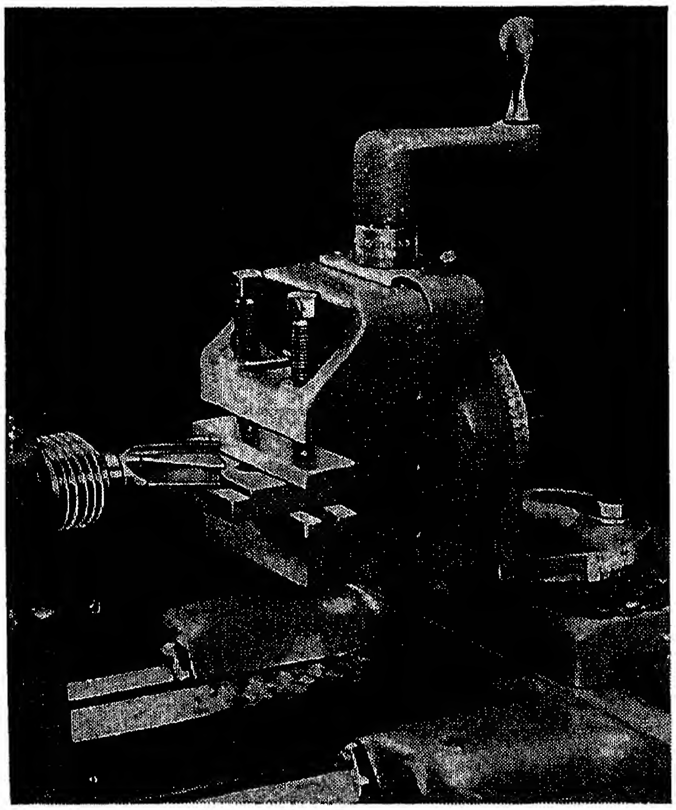


Fig. 5-20. A milling attachment on a lathe. (Courtesy South Bend Lathe Works.)

A *grinding attachment* or *toolpost grinder* is mounted on the cross-slide and carries a grinding wheel that is applied to the work in place of the usual cutting tool. One is shown in Fig. 17-1.

The *ball turning attachment* is a special kind of tool carrier that replaces the compound rest on a lathe to turn segments of spheres. The attachement revolves about a pivot, driven by hand through a worm and worm wheel. The tool bit is offset from center and is fed in a circular arc around the revolving workpiece.

A *carriage stop* makes it possible to set the carriage accurately and quickly in definite positions. Carriage stops are used for such operations as spacing grooves, facing a shoulder to a specific dimension, and cutting off a workpiece to a definite thickness. One kind of carriage stop, called a *micrometer stop*, is shown in Fig. 5-21. It has a micrometer spindle and dial that permit adjustments of 0.001 in. The carriage is positioned by being brought against the end of the micrometer spindle, held in a bracket clamped to the front way of the bed.

Another kind of stop is the *dial indicator stop*. It is also fastened to the bed but has a dial indicator instead of a micrometer spindle and gives visual evidence of the carriage position. This type of stop is accurate and quick in operation.

An *automatic stop* has a rail running along the front of the bed below the apron. Sliding dogs or collars can be positioned and clamped where desired along the rail. They are set to trip a lever that disengages the feed clutch and stops the carriage at desired positions.

A *turret* or *multiple* position stop carries four or more adjustable stop screws on a barrel. The barrel is indexed to bring each stop screw in place in turn to position the carriage

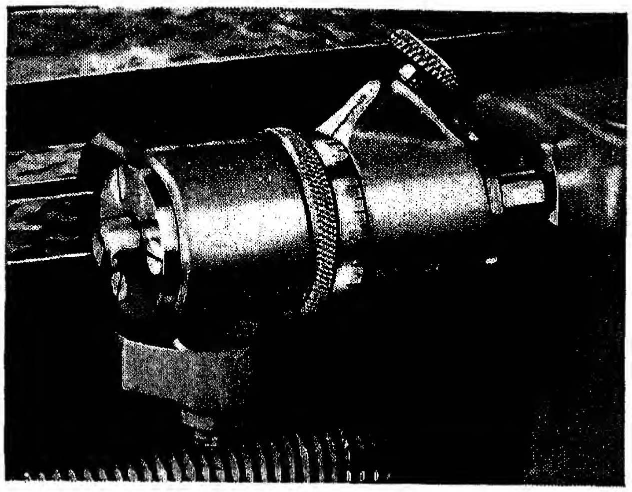


Fig. 5-21. A micrometer carriage stop. (Courtesy South Bend Lathe Works.)

Stops like those just described to position the carriage along the bed are also available for positioning the cross-slide to make the tool repeat in cutting desired diameters.

Lathes engaged in production often have a *square turret* in place of a toolpost. Some have a carriage and *hexagonal turret* in place of the tailstock. These are very much like the turrets on turret lathes, shown in Figs. 21-1 and 21-2. In effect, these attachments convert an ordinary lathe into a hand screw machine or turret lathe.

Multiple point tools like milling cutters and reamers have each tooth backed off behind its cutting edge for relief. The teeth may be backed off by means of a *relieving attachment* on a lathe. A short slide is placed on the carriage to carry the cutting tool, in place of the compound rest. This slide is connected to the headstock spindle through a chain of gears and a telescoping shaft and is made to move in and out at regular intervals as the spindle turns. As the workpiece revolves, the tool takes repeated in and out cuts over the surface.

The *rapid traverse attachment* on a lathe gives a rapid rate of power traverse to the carriage. It saves operator effort and time, particularly on long lathe beds.

Lathe manufacturers have developed and are able to furnish many other ingenious attachments to enable lathes to perform special kinds of operations economically at moderate and large-scale production rates. Examples of these are attachments for turning crankshafts and pieces with unusual shapes. The engineer should be aware that the experience and facilities of the machine tool manufacturers are always available for solution of his particular problems.

Questions

1. Name and describe the major units of lathes.
2. Sketch a back geared lathe spindle drive.

3. What movements does the tool on a lathe have? How are they obtained?
4. From where does the power feed come on a lathe? How is it varied?
5. How is the carriage moved for cutting threads on a lathe? For other operations?
6. How is the size of a lathe designated?
7. What is an engine lathe?
8. How does a toolroom lathe differ from an engine lathe?
9. Describe a duplicating lathe.
10. Describe a production lathe.
11. Describe an automatic lathe.
12. What are lathe accessories and attachments?
13. What are the features of a 3-jaw universal chuck? What disadvantage does it have?
14. What is a self-centering chuck?
15. How is a four-jaw independent chuck operated? What are its advantages and disadvantages?
16. What is a combination chuck?
17. For what purposes are air or hydraulic chucks used?
18. Describe a drill chuck.
19. What is a collet chuck? How is it used?
20. When must the tip of a center be lubricated?
21. What may be done to keep a dog from marring a workpiece surface?
22. How does a center rest differ from a follower rest?
23. Describe a taper attachment.
24. Of what advantage is a thread dial indicator?

References

How to Run a Lathe. South Bend, Ind.: South Bend Lathe Works.

Manufacturers' Catalogs:

American Tool Works Co., Cincinnati, Ohio.
Cincinnati Lathe Co., Cincinnati, Ohio.
Cushman Chuck Co., The, Hartford, Conn.
R. K. LeBlond Machine Tool Co., Cincinnati, Ohio.
Monarch Machine Tool Co., The, Sydney, Ohio.
South Bend Lathe Works, South Bend, Ind.
Sundstrand Machine Tool Co., Rockford, Ill.
Thomas Hoist Co., Chicago, Ill.
Wahlstrom Tool Division, American Machine and Foundry Co., Brooklyn, N.Y.
Warner and Swasey Co., Cleveland, Ohio.

Chapter 6

LATHE OPERATIONS

Lathe operations may be classified according to the ways that the work is held. In *plain or straight turning,* the workpiece is revolved between centers. *Chuck work* is done when the workpiece is revolved in a chuck or fixture or on a face plate. Most operations on the lathe call for the work to be rotated, but sometimes the workpiece is held still. It may be held in a vise or fixture on the carriage or by the tailstock. Then a cutter, such as a center drill or milling cutter, is revolved by the spindle.

Lathe operations may be described by the kinds of cuts taken. Common cutting operations are turning, facing, grooving, taper turning, threading, and knurling. These may be done either between centers or by chucking. Other work, such as drilling, boring, reaming, counterboring, internal threading, taper boring, and cutting off can be performed only if the workpiece is chucked, clamped, or fastened while revolved.

Plain or Straight Turning

A typical example of plain or straight turning between centers is given in Fig. 5-16. The piece has center holes in the ends for bearing surfaces. Work supported in this way can withstand heavier cuts than if held at just one end.

Centering. A good center hole gives a full bearing behind the point of the center as shown in Fig. 6-1 A. Too deep or too shallow

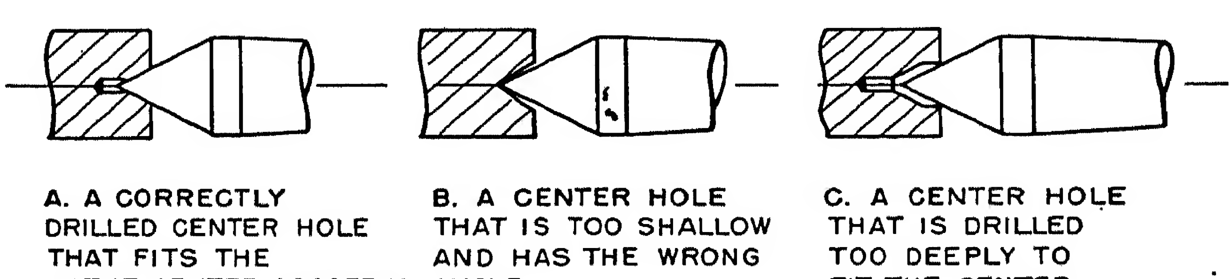


Fig. 6-1. Correct and incorrect center holes.

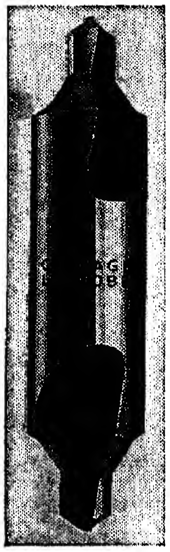


Fig. 6-2. A combination center drill and countersink. (Courtesy Chicago-Latrobe Twist Drill Works.)

a center hole results in an improper bearing and runout of the workpiece, as indiciated at B and C in Fig. 6-1. A center hole must be put in deeply enough so that it will not be removed when the end of the workpiece is faced.

Center holes are usually drilled with a combination center drill and countersink, depicted in Fig. 6-2. Several standard sizes are available. Suitable drill body sizes and hole sizes for work diameters up to 4 in. are listed in Table III.

Table III

Sizes of Center Drills and Holes for Various Diameters of Workpieces in inches.

Diameter of workpiece	*Large diameter of center hole*	*Diameter of tip of center drill*	*Diameter of body of center drill*	*Center drill designation*
3/16 to 5/16	1/8	1/16	13/64	C-2
21/64 to 1	3/16	3/32	3/10	E-1
1 1/64 to 2	1/4	1/8	3/10	E-2
2 1/16 to 4	5/16	5/32	7/16	F-1

Center drills are fragile and must be treated carefully, run at high speeds, and fed slowly to avoid breakage.

Center holes may be located and drilled in a number of ways. A workpiece may be held and rotated by a chuck on the headstock spindle of a lathe as shown in Fig. 6-3. The workpiece is first faced smooth before being centered. The center drill is held by a drill chuck in the tailstock spindle and fed into the work. A piece extending more than 10 times its diameter beyond the chuck should be supported by a center rest.

Fig. 6-3. A workpiece chucked on a lathe for drilling a center hole. (Courtesy South Bend Lathe Works.)

Centering machines are useful for centering work rapidly, especially in fairly large quantities. A common type of centering machine has a station-

ary chuck for holding the work and a high speed spindle for rotating and feeding the center drill.

If the work is held in a universal chuck on a lathe or centering machine as just described, the center hole is located automatically. By other methods, lines are scribed on the end of the workpiece to show the position of the center hole. Several ways of doing this are indicated in Fig. 6-4. The center head and scale of Fig. 6-4 B provide a fast and easy way of scribing two lines that intersect at the center of a round piece. A prick punch mark is made for each

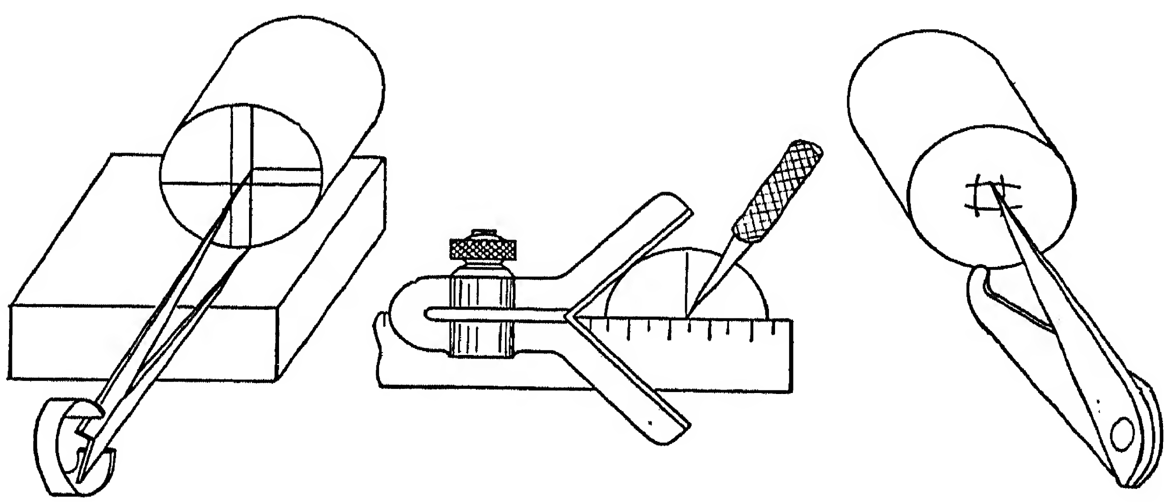


Fig. 6-4. Several ways of locating centers.

center, deep enough so that the workpiece can be supported between centers. The accuracy of the center punch marks may be checked by rotating the piece between dead centers and marking the high spots with chalk. The center punch marks may be driven toward or remarked nearer to the high side, and then the piece is rechecked. When the center marks are correct, they are deepened to give a good start to a center drill.

A piece may be center drilled on a lathe by holding it by hand with one center punch mark on the dead center in the tailstock. The tailstock spindle is fed forward, and the other center punch mark is brought up to a center drill in a chuck revolved by the headstock spindle. Center holes may also be drilled in a drill press. If a combination center drill and countersink is not available, a small twist drill may be used, followed by a 60° countersink.

Setup. In preparation for turning between centers, the tapered

holes in the spindles, tapered sleeves, and centers must be wiped clean to remove any small chips or particles of dirt that would cause the centers to run out of true. Centers are then placed in the headstock and tailstock spindles. If a center is of the right size, it will stick when pushed in place. If the center taper is too small a tapered sleeve that fits the spindle hole is slipped over it. A dog plate is put on the headstock spindle.

The centers of a lathe must be in line to turn a straight piece.

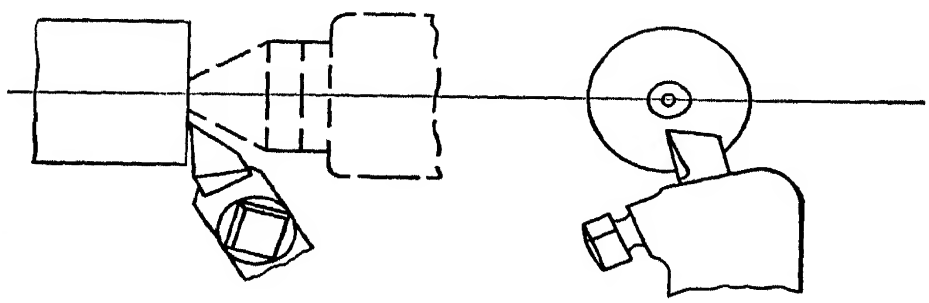


Fig. 6-5. The position of the cutter for facing the end of a workpiece.

The centers may be checked for alignment by moving the tailstock forward until the points almost touch. A more accurate method is to place a bar known to be straight between centers and run an indicator held by the toolpost along the side of the bar. If the centers are out of line, the condition may be corrected by adjusting the tailstock in the way described later in this chapter for the set-over method of taper turning. The adjusting means are illustrated in Fig. 6-28.

A lathe dog is clamped on one end of the workpiece, the tailstock is positioned to accommodate the workpiece and clamped in place, and the workpiece is placed between centers. The tail of the dog must enter a slot in the dog plate but must be free and not bind in the bottom of the slot. A drop of oil or white lead in oil should be put on the tip of the dead center or in the center hole. The tailstock center is brought up until it makes positive contact with the center hole of the workpiece. The centers must not bind the work tightly because the heat generated in a tight bearing when the workpiece turns can easily ruin either the center or workpiece. The tailstock spindle is clamped to hold the center in position. When metal is cut between centers, it becomes hot and often expands enough to cause binding. The tailstock center must then be readjusted. That may have to be done several times in machining a long shaft.

Facing the ends. Before any diameters are turned on a piece, its ends should be faced square and to the required length. A right-hand facing tool, described in Fig. 4-10, is mounted in a toolholder in the toolpost. The cutting edge should be set at the same height as the center of the workpiece. That may be done by setting the tool point to the same height as the tailstock center point.

A spindle speed is selected to give the proper surface speed at the outer edge of the face, and the lathe is started. The tool is brought in to clean stock from around the center for the desired depth of cut and then is fed outward, generally by hand. Care must be taken not to break the point of the tool against the tailstock center. For a true face, the saddle is clamped to the bed for the finishing cut. The flatness of the face may be checked by placing a straightedge or the edge of a scale against it.

To face the other end, the dog is changed and the piece turned end to end between centers. Firm joint calipers can be used to measure the length of the workpiece.

A facing tool is mounted in a toolholder on the toolpost in Fig. 6-3 for facing the end of the piece held in a chuck.

Turning. A right-hand turning tool, like the one in Fig. 4-10, is most commonly used for turning diameters because it can be fed toward the headstock which is more rugged and can take the thrust better than the tailstock. The nose of the tool should be set on center or not over 1/16 in. above center, depending upon the diameter of the workpiece and the shape of the tool. A tool too far above center loses the benefit of front relief and rubs. For a heavy cut, the toolholder is turned so that the point of the tool trails slightly, as shown on Fig. 5-16. That is done so that the tool will swing away from the work and not dig in if the side thrust gets large enough to push the tool aside. For light cuts, the tool point may be set ahead of the compound rest, as illustrated in Fig. 6-6, especially if a cut must be taken near a dog or chuck. That reduces the chance of running the compound rest into the revolving dog or chuck and having a wreck.

The spindle speed of the lathe is selected to suit the diameter and material being turned. The rate of feed and depth of cut for roughing should be as heavy as the tool, workpiece, and machine will stand. Typical speeds, feeds, and depths of cut are suggested in Chapter 4. A diameter usually is rough turned in one or more cuts

to within 1/32 in. of size to remove stock in the shortest possible time. Finishing cuts should be light and made at slow feeds for accuracy of size and smooth surface finish. Sometimes a heavier tool is used for roughing, but generally the roughing and finishing tools are the same kind.

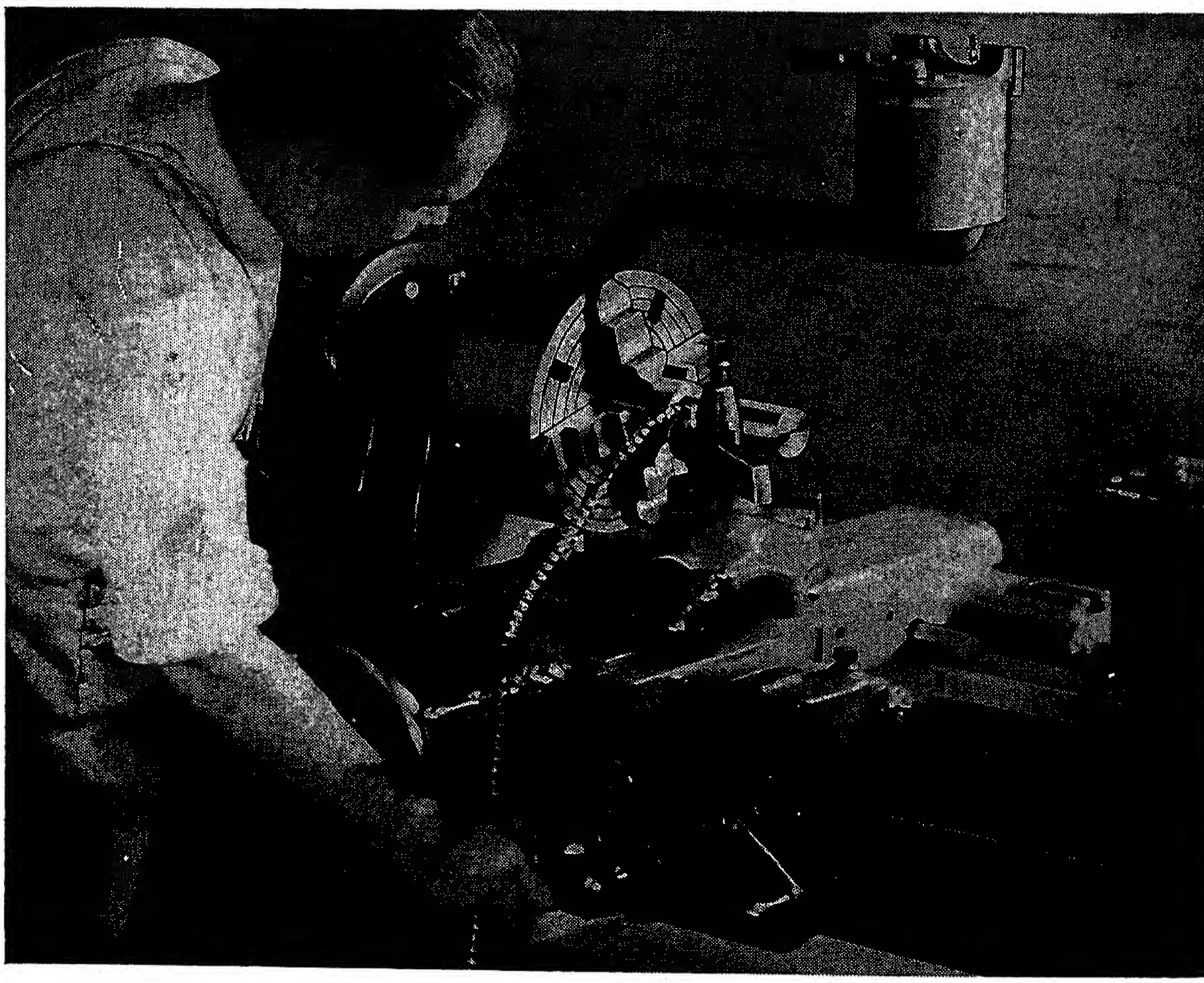


Fig. 6-6. A shaft being turned while gripped in a four jaw chuck. The tool is set forward to avoid interference between the compound rest and chuck jaws. (Courtesy South Bend Lathe Works.)

A turning tool may be brought up to touch the outside of the workpiece to establish its first setting. Then the tool is traversed past the end of the workpiece and adjusted for the desired depth of cut by the micrometer dial on the cross-feed screw. Also, after a tool has finished a cut at a certain setting, it may be set in for its next cut bv the dial. At the start, a short cut of about ¼ in. is taken,

and the diameter is checked. If the size is not right, the tool is returned to the starting point and its setting readjusted before continuing with the cut. Care must be taken that the cross screw and dial are turned always in the same direction when making successive adjustments. Otherwise the dial readings will not be correct because if the screw is reversed, backlash always causes an error. This must be watched in making adjustments for size on all machine tools.

A piece turned between centers over most of its length may be checked for straightness by measuring diameters near each end of the cut. The headstock and tailstock centers probably are out of line if the readings are not the same, and the tailstock may have to be adjusted. A lathe that is not properly leveled will not turn straight work.

A long and slender workpiece may spring away from a cutting tool. That causes chatter, inaccurate size, and may even bend a frail piece. The situation may be helped by taking light finishing cuts with a sharp tool. Where a piece is too likely to deflect, a steady rest, follower rest, or both may be put on a lathe to support the work as shown in Fig. 5-18. The jaws of a steady rest must bear on a surface that runs true. A light cut a little wider than the jaws is made on the workpiece and is called a spot. The jaws are adjusted to make contact with the spot to hold the piece firmly but not prevent it from turning easily. A follower rest is mounted on the carriage, and its shoes follow the cutting tool and bear on the newly cut surface.

The tolerances that can be held in turning depend upon the size of the dimensions, the condition of the machine, and the skill of the workman. Under average conditions and performances, reasonable tolerances for rough turning range from 0.005 in. for diameters under ½ in. to 0.015 in. for diameters over 2 in. For finish turning, tolerances from 0.002 in. for diameters below ½ in. to 0.007 in. for diameters above 2 in. are usual. It must be emphasized that smaller tolerances than these can be and are held on lathes, but the smaller they are, the higher is the cost of maintaining them.

Diameters may be measured or compared with an outside spring caliper if they are not required to be accurate within 1/64 in. Closer measurements are made with a micrometer caliper. Diameters may be checked in production with snap gages.

The length to which a diameter is to be turned may be marked

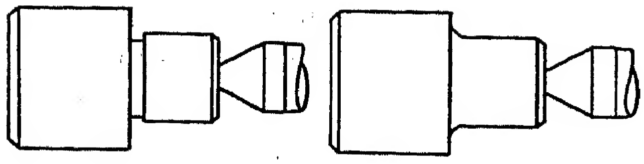


Fig. 6-7. An undercut and fillet.

by means of a hermaphrodite caliper. Chalk is applied around the work surface. The caliper is set to the required length, the hooked leg is placed against the face at the beginning of the cut, and a line is scribed around the revolving workpiece with the sharp leg. Lengths often are scaled. For closer tolerances, verniers and micrometers are used. For instance, the distance between two shoulders may be checked with a vernier caliper, and a distance across two shoulders by a micrometer caliper. A dimension cannot be machined more closely than it can be measured or gaged.

A shoulder with a fillet at the end of a diameter, like Fig. 6-7 B, may be finished with a round nose tool. When an undercut is required, as shown in Fig. 6-7 A, a groove may be cut to the desired depth with a cut-off tool before the small diameter is turned.

Chuck Work

Work is held by a chuck, collet, face plate, or fixture when it cannot conveniently be held between centers or when drilling, boring, tapping, or reaming must be done to it. Either the workpiece or tool may be revolved for internal operations. The workpiece generally is revolved on a lathe, but the opposite is done at times if it is more convenient to rotate the tool. Figure 6-8 is an example of a drill rotated by the headstock spindle for putting a hole in a stationary workpiece.

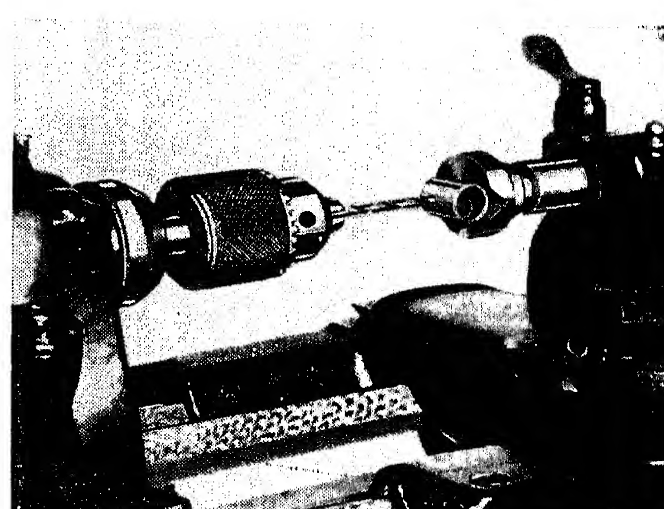


Fig. 6-8. Drilling an oil hole in a bushing held by a crotch center on the tailstock. (Courtesy South Bend Lathe Works.)

Conventional machines specifically for drilling, boring, tapping, and reaming will be described in later chapters. However, many such jobs can be and are done readily and quickly on lathes.

Work set-up. Before a chuck or other device is mounted on a lathe spindle, all mating surfaces must be wiped off and thor-

oughly cleaned because even a very small chip or particle will make the locating device run untrue.

A chuck or face plate should be set on a board across the bed ways when put on or taken off the spindle to prevent damage to the ways. A board of the proper thickness to position the chuck in line with the spindle and with grooves to match the V-ways can be very helpful in managing a heavy chuck. If the spindle is threaded, a chuck or face plate should be screwed on slowly by hand. Turning the spindle by power is likely to jam the chuck and make it difficult to remove. In any event, a chuck is usually made tighter on a threaded spindle nose by cutting torque. To loosen

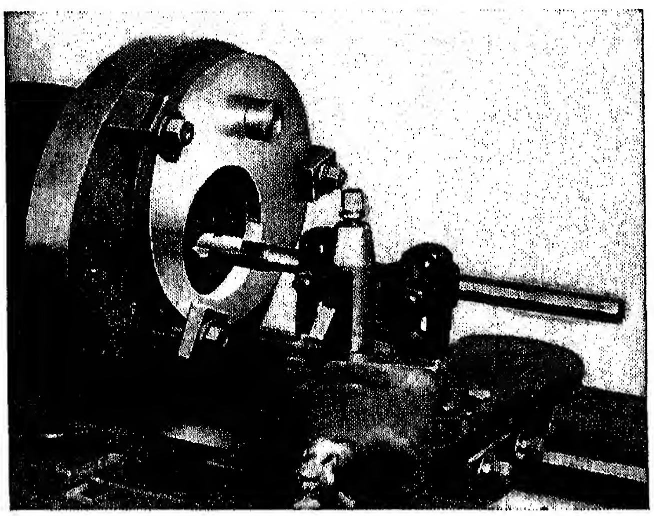


Fig. 6-9. A workpiece clamped to a faceplate for boring an eccentric hole. (Courtesy South Bend Lathe Works.)

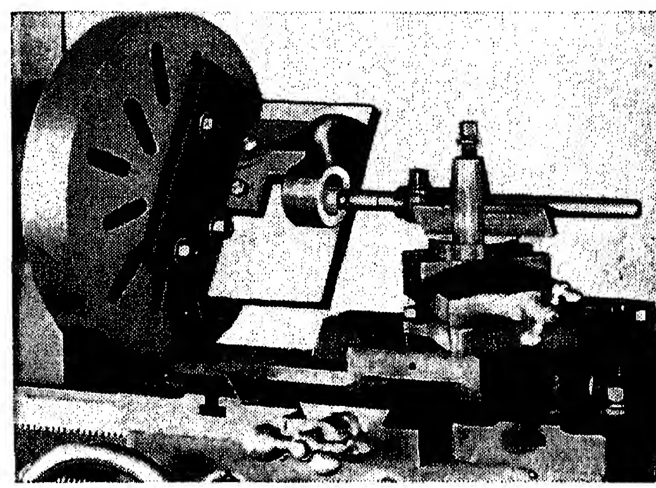


Fig. 6-10. Boring a bracket mounted on an angle plate attached to a faceplate. (Courtesy South Bend Lathe Works.)

a chuck to remove it from a threaded spindle, a wooden or aluminum block is placed on the back way of the bed under one of the jaws, and the spindle is turned backward by hand. On many lathes, chucks are held on by threaded sleeves, bolts, or cam locks and are removed by loosening the fasteners.

A workpiece is automatically centered in a universal chuck subject to the inherent inaccuracy of the chuck. For closer concentricity, a workpiece may be adjusted in an independent chuck to run as true as desired. Concentric rings are marked on the face of an independent chuck as approximate guides for centering round work. These rings are visible on the face of the chuck in Fig. 6-6. A piece of chalk may be held to rub against and mark the high side of a revolving workpiece. The chuck is then stopped, the jaw

opposite the chalk mark is loosened slightly, and the opposing jaw is tightened. The test is repeated until the work runs as true as desired. All four jaws must be tightened securely before any cut is taken.

A dial indicator may be applied to a workpiece in a chuck, as shown in Fig. 3-19, and adjustments made until the pointer on the dial shows the indicated surface to be running true. Runout of less than 0.001 in. can be realized by this method.

A workpiece clamped or bolted to a face plate, as illustrated in Figs. 6-9 and 6-10, can be indicated and adjusted to run as desired if the clamps are not fully tightened at first. After the desired surface has been positioned, the clamps are tightened securely before cutting. If most of the weight of a workpiece or fixture is on one side of a face plate, a counterweight may be attached to the other side.

The contact surfaces of the jaws of a universal chuck may be trued for close work. Soft jaws are turned or bored, hard jaws ground. While the jaws are being trued, they are made to grip a disk or ring of the same diameter as the workpiece to be held to simulate operating conditions.

Drills, Boring Tools, and Reamers

Twist drills. Drills are capable of putting holes in solid stock in contrast to boring tools and reamers which are suitable only for enlarging holes. The twist drill is the most common type of metal working drill. It is characterized by helical grooves or *flutes*, as depicted in Fig. 6-11. This form is obtained in some cases by twisting but the helical grooves of most drills are milled from the solid. Drills that have two flutes and are called *two lipped drills* are capable of piercing solid metal but are also used to enlarge holes. The latter operation is *counterdrilling*. Similar tools that have three or four helical flutes are sometimes called *core drills*. They are not capable of initiating holes, only of enlarging holes previously drilled or cored, and for that reason are more properly classified as boring tools or reamers. Often they are given names like *spiral reamers* or *core reamers*. Some are made with replaceable points.

The fluted portion of a drill is its *body*. The *point* is that portion of the end where the cutting is done. At the other end of the body is the *shank* where the drill is held and driven. Some drills have

straight shanks, others tapered shanks, as shown in Fig. 6-11. Straight shank drills are held by drill chucks. Tapered shank drills commonly have Morse tapers with driving tangs on their ends.

Drill sizes and materials. The size of a drill designates the nominal diameter of its body and the hole it is intended to produce. Standard drills are available in *numbered, lettered, fractional inch,* and *millimeter* sizes.

Numbered drills range from No. 80 (0.0135 in. diam) to No. 1 (0.2280 in. diam), and lettered drills from A (0.234 in. diam) to Z (0.413 in. dia). They are straight shank or *wire-type drills* for holes that need to have other than standard fractional sizes. A common use of numbered or lettered drills is to drill holes for tapping. For

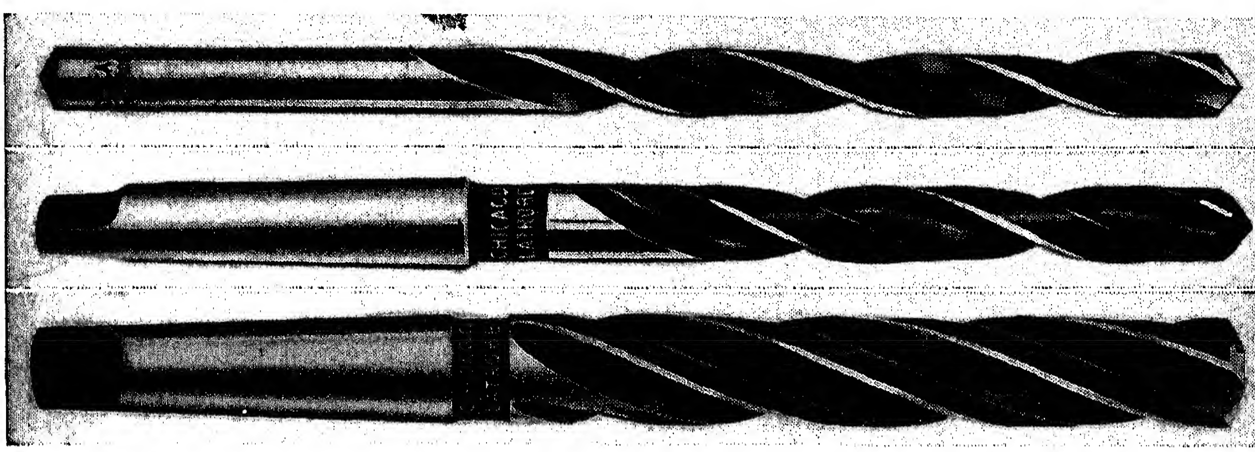


Fig. 6-11. (top) **A straight shank two lipped twist drill.** (middle) **A tapered shank two lipped twist drill.** (bottom) **A four flute core drill. (Courtesy Chicago-Latrobe Twist Drill Works.)**

example, a standard ¼ – 20 NC (see screw thread standards in Chapter 7) thread has a major diameter of about ¼ in. A hole to be tapped with that thread is drilled with a No. 7 (0.2010 in. diam) drill to leave enough stock for approximately 75 per cent of a full thread.

Fractional size drills range from 1/64 in. to 1¾ in. diameter by 1/64 in., to 2¼ in. diameter by 1/32 in., and to 3½ in. diameter by 1/16 in. steps. They are made with both straight and tapered shanks. Some twist drills are made as large as 6 in. diameter. Millimeter drills are available from 3 mm (0.1188 in. diam) to 77 mm (2.9921 in. diam) in ½ mm steps but are not common in the United States.

The size of a drill is generally stamped in the shank just behind the flutes. Drill lengths vary with diameter and many sizes are available in short and long lengths.

Carbon tool steel drills have a low first cost and are economical

for occasional usage but must be run at low cutting speeds. High speed steel drills are the most popular for production. They are usually stamped with the letters HS or HSS near the size marking. Drills tipped with cemented carbide are used for drilling hard material and for high speed operations. Tips on drills cannot be fully reinforced and are not favored for ordinary work.

Drill angles and edges. The body and point of a drill must have certain features for efficient performance. These are desig-

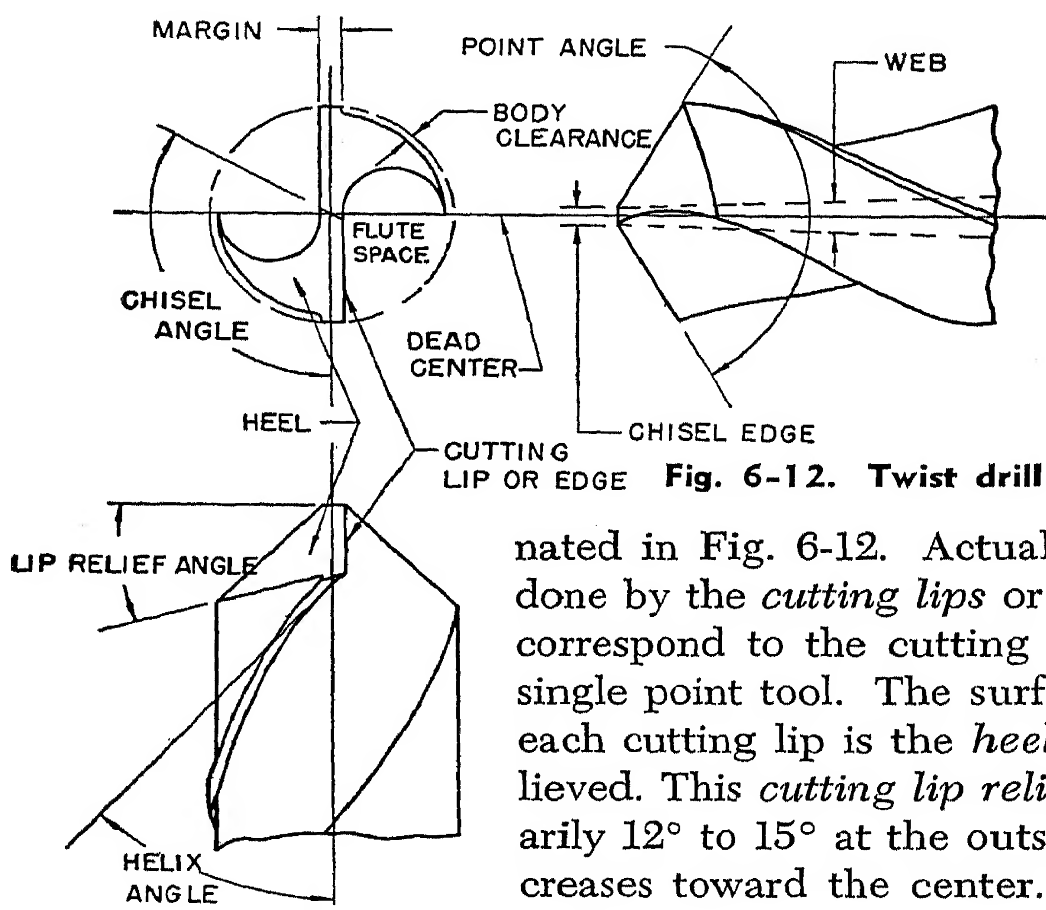


Fig. 6-12. Twist drill elements.

nated in Fig. 6-12. Actual cutting is done by the *cutting lips* or *edges* that correspond to the cutting edges of a single point tool. The surface behind each cutting lip is the *heel* and is relieved. This *cutting lip relief* is ordinarily 12° to 15° at the outside and increases toward the center. The *helix angle* of the flutes corresponds to the rake of the single point tool and ranges from 18° to 45° for general work. The smaller angles are suitable for hard materials, the larger angles for soft materials. A helix angle of about 30° is most often used.

The point of a drill is tapered with a *point angle* between the cutting edges. That angle is 118° for average work but has different values for certain specific purposes. For example, a point angle of 136° is recommended for hard manganese steel but an angle of only 60° for wood and fiber. The cutting edges must lie at equal angles from the axis of the drill. For a point angle of 118°, each edge must be 59° from the axis.

The *chisel edge* is the sharp edge at the extreme tip of the point between the cutting edges. The large angle between the chisel edge and each cutting edge is normally 135°. The chisel edge is the end of the section between the flute spaces called the *web.* The web is the center section or backbone of the drill. It increases in thickness toward the shank. Consequently, the chisel edge becomes wider as the drill is shortened by repeated sharpening, and more thrust is required to drive it through the material. To keep down the thrust, the web of the drill is thinned by grinding the bottoms of the flutes behind the chisel edge.

The bodies of all but very small drills are relieved except for a narrow *margin* or strip along the length of each flute. This relief is called *body clearance.* The full diameter of the body is measured across the margins. The body clearance reduces rubbing between the drill and hole and allows cutting fluid to reach the point of the drill. Also, the body decreases a few thousandths of an inch in diameter from the point to the shank to keep down rubbing.

A twist drill is sharpened by grinding the heel behind the cutting edges or lips on the point. Skilled mechanics may grind drills off hand with passable results, but generally the best results are obtained on drill grinding machines described in Chapter 11. When a drill is sharpened, it is important that its original correct features be reproduced faithfully. Otherwise the drill cannot be expected to have a reasonable life nor produce accurate holes. Sufficient cutting lip clearance must be provided to keep the heel from rubbing as the drill is fed into the work. Too small an angle may cause enough drag to split the drill up the web. On the other hand, too much lip clearance removes too much metal from the heel. The cutting edge is weakened and may chip away.

The edges of a drill pursue helical paths as the drill is revolved and fed. The angle of helix is larger near the center than at the outside. The clearance angle must correspond to the helix angle of the cut.

The cutting edges or lips must both be at the same angle with the axis of the drill and of the same length. If either the angles or lengths or both are different, the drill cuts a large and irregular hole and receives severe punishment. The results of these faults are depicted in Fig. 6-13.

Drill speeds and feeds. The speed at which a drill should be run

depends upon the work material, cutting fluid, and other factors discussed in Chapter 4. In general, the proper peripheral surface cutting speed of a drill is about the same as for a single point tool under comparable circumstances.

The feed of a drill is the distance in inches it advances in one revolution. Feeds depend largely upon the size of the drill and the work material. As a general guide, the Cleveland Twist Drill Co. recommends "a feed of 0.001 to 0.002 in. per rev. for drills smaller than ⅛ in., 0.002 to 0.004 for drills ⅛ to ¼ in., 0.004 to 0.007 for drills ¼ to ½ in., 0.007 to 0.015 for drills ½ to 1 in., and 0.015 to 0.025 for drills larger than 1 in. Alloy and hard steels should generally be

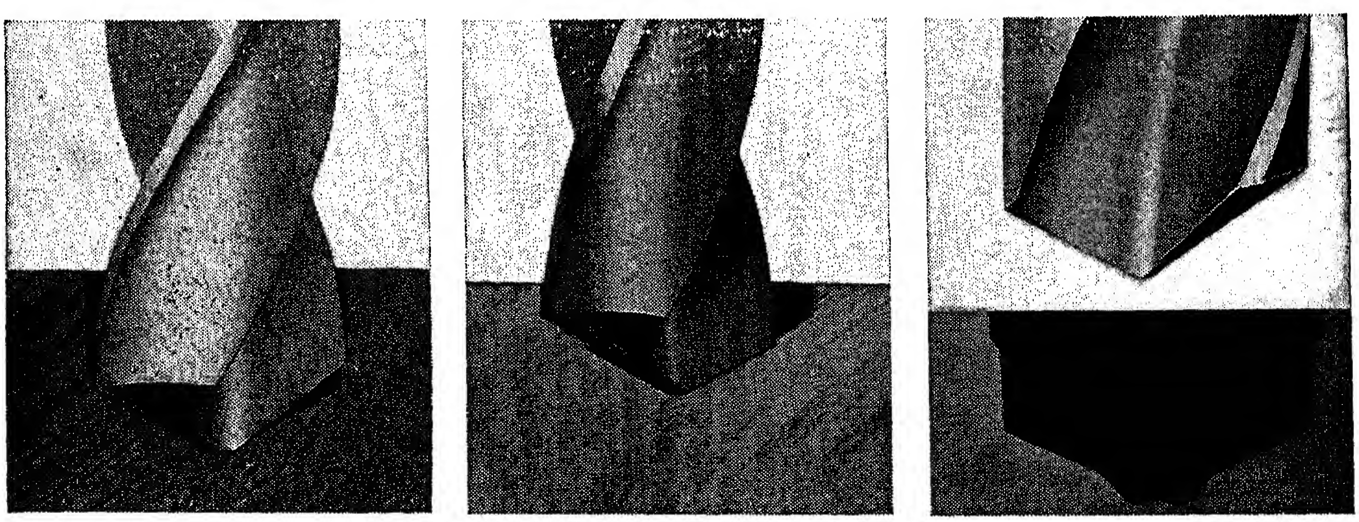


Fig. 6-13. The results of poor drill grinding. (left) **Lips ground at different angles.** (middle) **Lips ground at same angles but of different lengths.** (right) **Both angles and lengths of lips are different. (Courtesy Cleveland Twist Drill Co.)**

drilled at a lighter feed than given above while cast iron, brass, and aluminum may usually be drilled with a heavier feed than given above."

Boring tools. Single point tools are most commonly used for boring on the lathe, particularly for general purpose work. A single point boring tool is ground like a left-hand turning tool except that the front clearance angle is larger to avoid rubbing on the concave work surface. The boring tool is applied to the operator's side of the hole. Speeds and feeds for boring are comparable to those for turning.

The point of a boring tool may be forged on the end of a shank like the one in Fig. 6-22. That form is common for small sizes. For moderate-size and large holes, a bit is usually mounted in a boring

bar, as shown in Figs. 4-10 and 4-11. Two or more bits may be carried by one bar for production work. They may serve for successive rough and finish cuts or for boring more than one diameter at one time. One or more blades may be added for facing.

Multipoint or multiblade tools are used for boring. Several blades acting at once help support the tool in the cut and keep

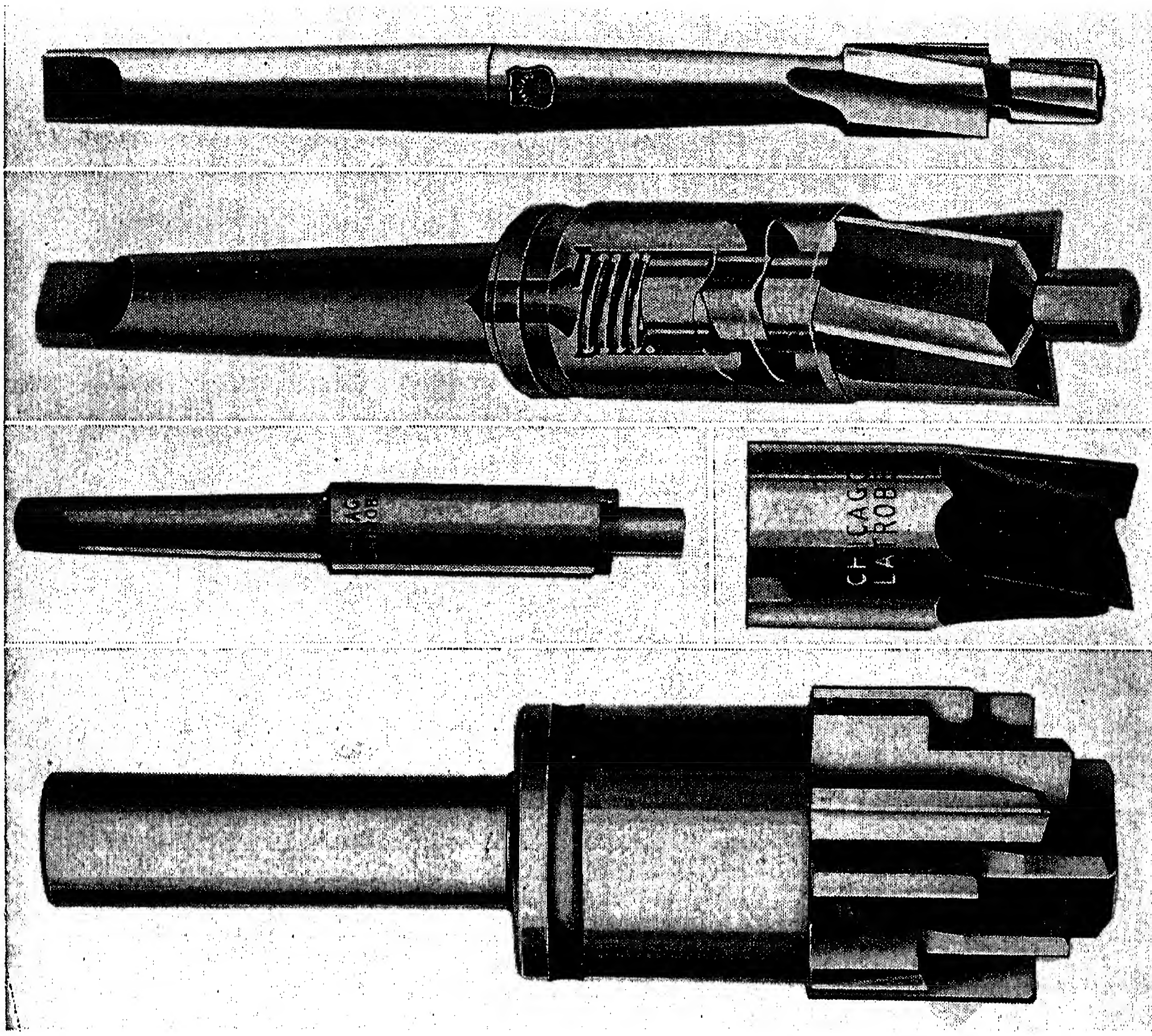


Fig. 6-14. Typical boring and counterboring tools. A. A solid taper shank counterbore with integral pilot. (Courtesy The Standard Tool Co.) B. An interchangeable counterbore. The cutter has a short shank with two driving lugs. It is inserted in the tapered shank holder and locked in place with a one quarter turn, as indicated by the phantom view. The cutter can easily be removed from the driver and replaced with another. The pilot is replaceable. (Courtesy Ex-Cell-O Corp.) C. A replaceable counterboring tool and its holder. The cutter fits over the small diameter and is driven by the keys on one end of the driver. (Courtesy Chicago-Latrobe Twist Drill Works.) D. A special multi diameter boring cutter for machining 8 surfaces in one operation. (Courtesy Eclipse Counterbore Co.)

down chatter. Core drills and end mills are used for boring. Among the multiblade boring tools are *counterbores* or *counterboring tools.* They may have integral or inserted blades and often have pilots ahead of their cutting edges to locate and guide them. They may have full length tapered shanks, stub shanks to fit holders, or axial holes for mounting on arbors. A solid counterbore with a full length tapered shank and integral pilot is shown in Fig. 6-14 A. Counterbores are often used to enlarge holes to take the heads of cap and machine screws. Two kinds of interchangeable counterbores are illustrated in Fig. 6-14 B and C. The bodies having the cutting edges can be replaced by others of the same size or different sizes. Counterboring is done at about the same speeds and feeds as drilling, sometimes slightly less.

Counterboring tools are also used to take light cuts to finish the surfaces around holdes. That is *spotfacing,* and in that role the tools are called *spotfacers. Countersinking tools* or *countersinks* are similar to counterboring tools except that their blades are ground at an angle to chamfer the ends of holes. Included angles of 45°, 60°, 82°, and 90° are common. Whereas a combination center drill and countersink may serve to form a countersink for a hole before it is drilled, a regular countersinking tool makes the countersink after the hole is drilled and generally has a pilot, like a counterbore, to guide it true with the hole.

An example of a special boring tool, one for machining eight surfaces in one operation, is given in Fig. 6-14 D. Staggered blades for the various cuts facilitate grinding to resharpen the edges.

Straight reamers. Reamers in some respects might be called boring tools. They have two or more blades and are used to enlarge holes, but generally they are confined to removing relatively small amounts of stock. The outstanding feature of reamers is that they are capable of giving better finishes and accuracy than are normally obtainable from drills and boring tools.

Reamers are made in a number of styles to suit various kinds of operations and purposes. They may be hand or machine driven, for roughing or for finishing work, have integral shanks or be attached to holders, be solid or have inserted blades that may be expanded or adjusted, have straight or helical flutes, and have straight, tapered, or other shapes.

A *hand reamer* has a straight shank with a square tang for a

wrench as shown in Fig. 6-15 A. It may have straight or spiral flutes and be solid or expansible. In any case, a hand reamer is expected to remove only a few thousandths of an inch of metal from a hole. The teeth are ground with relief behind their cutting edges along their lengths. The reamer is tapered from 0.005 to 0.010 in. for the first third of the length of its flutes for starting, is straight and to size over the next third, and usually falls off to about 0.005 in. undersize at the shank.

Machine or *chucking reamers* are made with or without relief, with straight or spiral flutes, and solid or with inserted blades. Common forms are described in the paragraphs that follow.

A *rose chucking reamer* is cylindrically ground and has no relief behind the outer edges of the teeth. It cuts on the end chamfer of the teeth. A solid rose chucking reamer with a tapered shank and straight flutes is shown in Fig. 6-15 B. Rose reamers are used for heavy roughing cuts, particularly for cleaning out cored holes, and not for especially smooth holes.

Two kinds of machine reamers with relief behind the outside edges of their teeth, as well as a chamfer on the front of the teeth, are taper-shank jobbers' reamers and fluted chucking reamers. The *jobbers' reamer* has long flutes but is not tapered like a hand reamer. It cuts on the chamfer on the front of the teeth. Its shank is tapered for machine use. A *fluted chucking reamer* has shorter flutes and relief behind all edges of the teeth. Fluted chucking reamers are

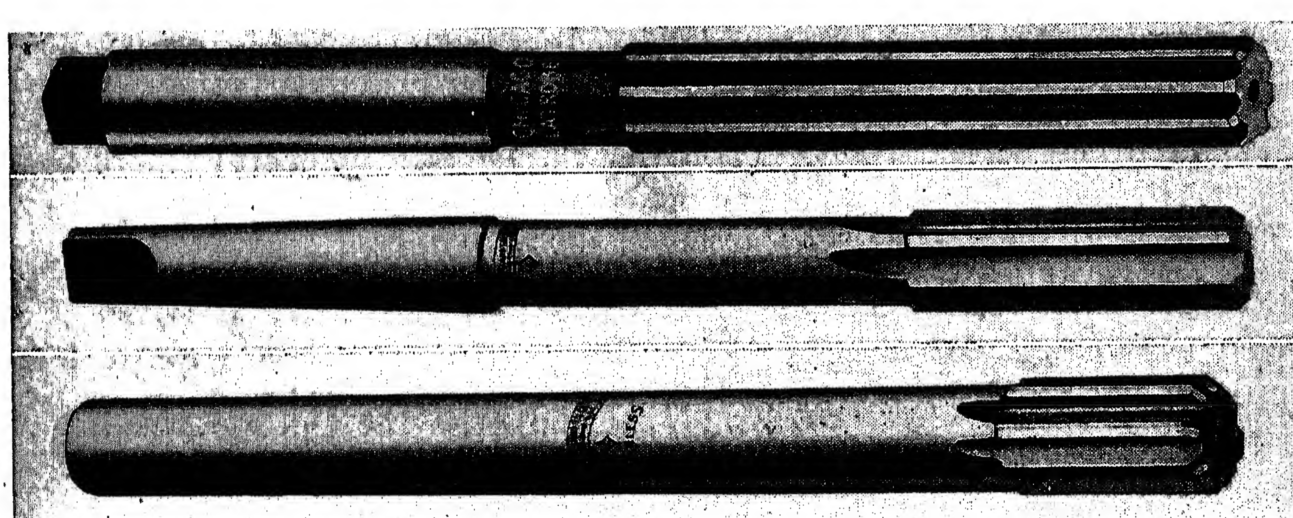
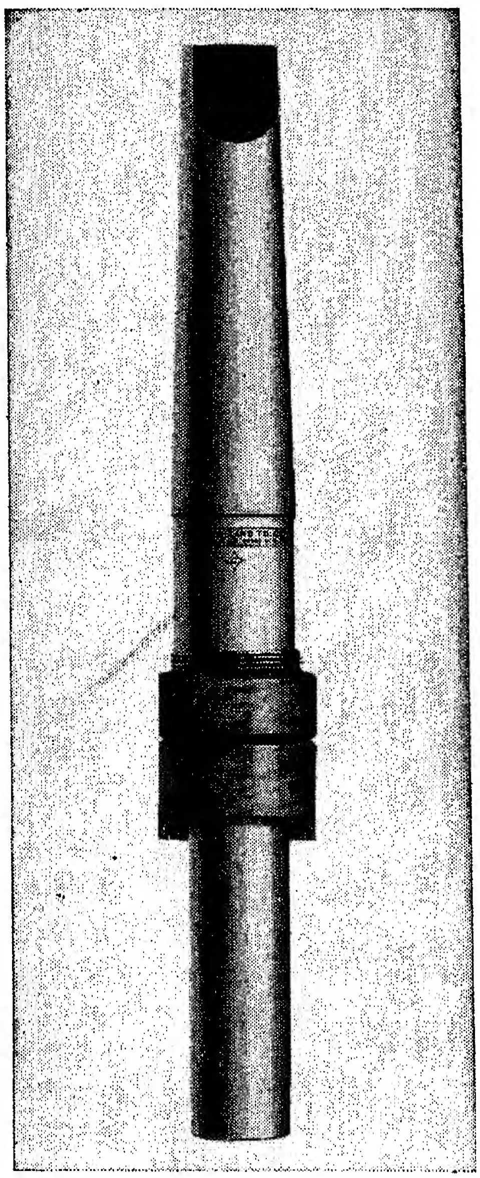


Fig. 6-15. Typical reamers. A. A straight flute solid hand reamer. (Courtesy Chicago-Latrobe Twist Drill Works.) B. A rose chucking reamer with tapered shank. (Courtesy Cleveland Twist Drill Co.) C. A straight flute expansion chucking reamer with tipped blades. (Courtesy Cleveland Twist Drill Co.)

designed for light finishing cuts and give best results when not held rigidly but allowed to float to seek alignment with a hole.

Reamers over ¾ in. diameter are made as shells to save cutting tool material. A typical *shell reamer* is illustrated in Fig. 6-16.



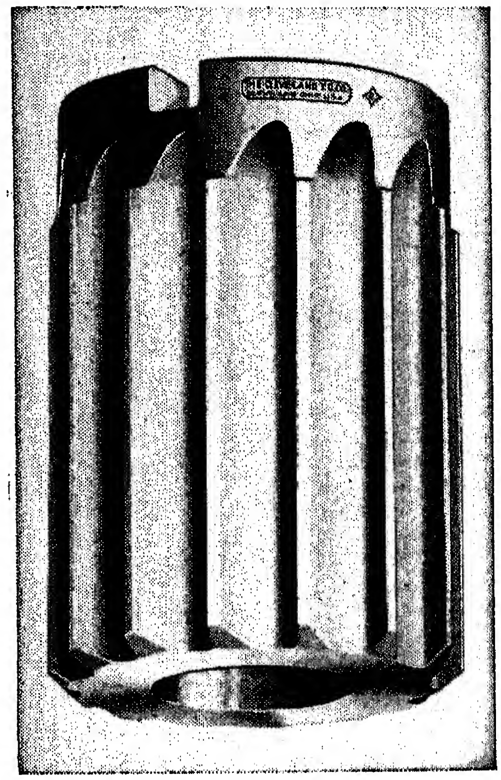


Fig. 6-16. A fluted shell reamer and arbor. (Courtesy Cleveland Twist Drill Co.)

Shells may be ground as rose or fluted reamers. They have holes with a slight taper that sticks on an arbor, like the one in Fig. 6-16. Keys on the arbor fit slots in the shell for driving. An arbor can take a variety of sizes of shells. When a shell is worn out, it is replaced on the arbor.

An *expansion reamer* can be enlarged to remove an extra few thousandths of an inch from a hole or to compensate for wear. Both hand and machine reamers are made in expansible models. One of a number of designs of expansion reamers is shown in Fig. 6-15 C. The body is hollow and has several slots running part way along the flutes. A tapered and threaded pin expands the body. Expansion of only 0.005 to 0.015 in. is feasible, depending upon the size of the reamer.

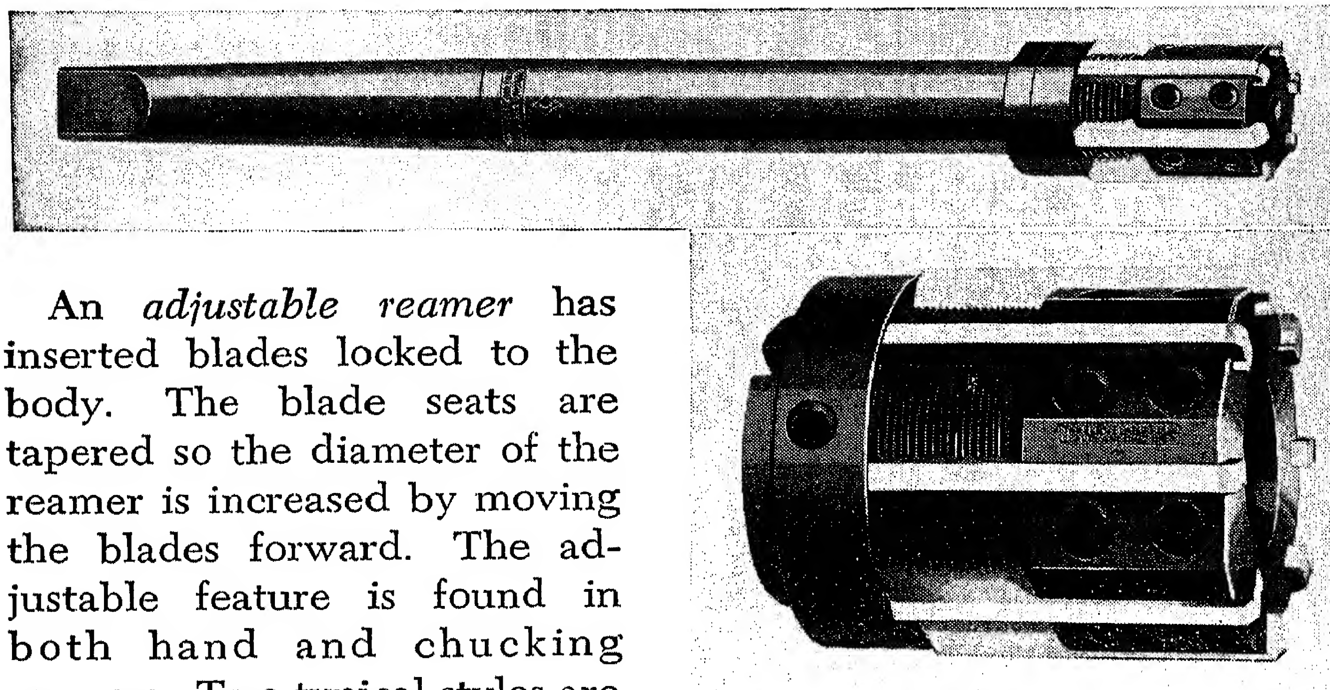


Fig. 6-17. Adjustable reamers. (top) A taper shank adjustable chucking reamer. (bottom) An adjustable shell reamer. (Courtesy Cleveland Twist Drill Co.)

An *adjustable reamer* has inserted blades locked to the body. The blade seats are tapered so the diameter of the reamer is increased by moving the blades forward. The adjustable feature is found in both hand and chucking reamers. Two typical styles are shown in Fig. 6-17. The blades can be ground a number of times and when used up, new ones are inserted. Reamers below 1 in. diameter usually are solid, from 1 in. to 3 or 4 in. may be solid or have inserted blades, but above those sizes almost always have inserted blades.

The cutting angles of a reamer are designated in Fig. 6-18. The amount of rake varies with conditions. Hand reamers have about 1° negative rake. Machine reamers are given no rake for cast iron and bronze, 5° to 7° positive rake for steel, and 7° to 10° for aluminum. The chamfer on the ends of the blades of machine reamers does most of the cutting. Some reamers are given a secondary chamfer to clean up the hole and produce a better finish. Some reamers have a small cylindrical margin; others do not. A reamer may have three kinds of relief. One kind is a relief of 6° to 8° behind the cutting edge of the chamfer. Another is a relief behind the peripheral edges that

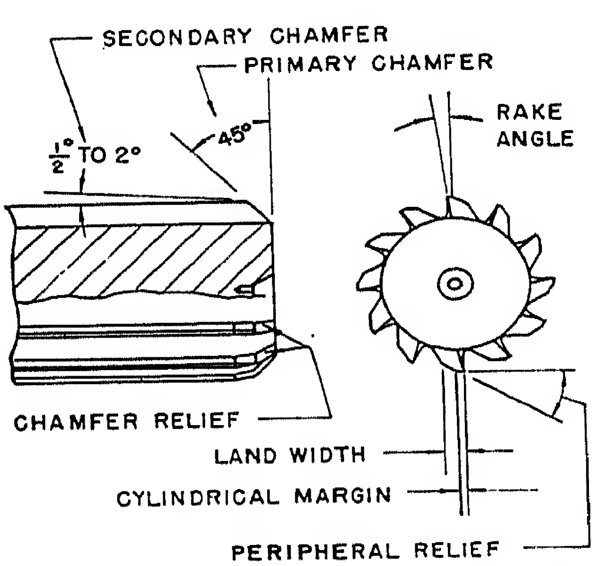


Fig. 6-18. Machine reamer angles.

helps relieve the radial pressure on the reamer in the hole and reduces rubbing. The third relief is provided by a 0.005 in. longitudinal back taper of the teeth on many reamers.

Reamers usually but not always have an even number of flutes. The number ranges upward from four for small diameters. The blades often are spaced at irregular intervals to reduce chatter. Reamer blades are commonly made of carbon tool steel or high speed steel. Carbide tips often are put on production reamers.

Reamers with straight teeth commonly are used for general purpose work. Many opinions have been expressed regarding the merits of reamers with right- or left-hand helix angles. Generally, left-hand helix angles are used for finish reaming, and right-hand angles for roughing. However, good results have been obtained in speed and finish from right-hand reamers for finishing. A style

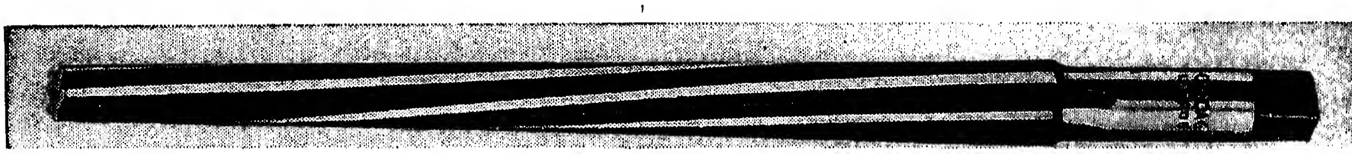


Fig. 6-19. A straight flute taper pin reamer. (Courtesy Chicago-Latrobe Twist Drill Works.)

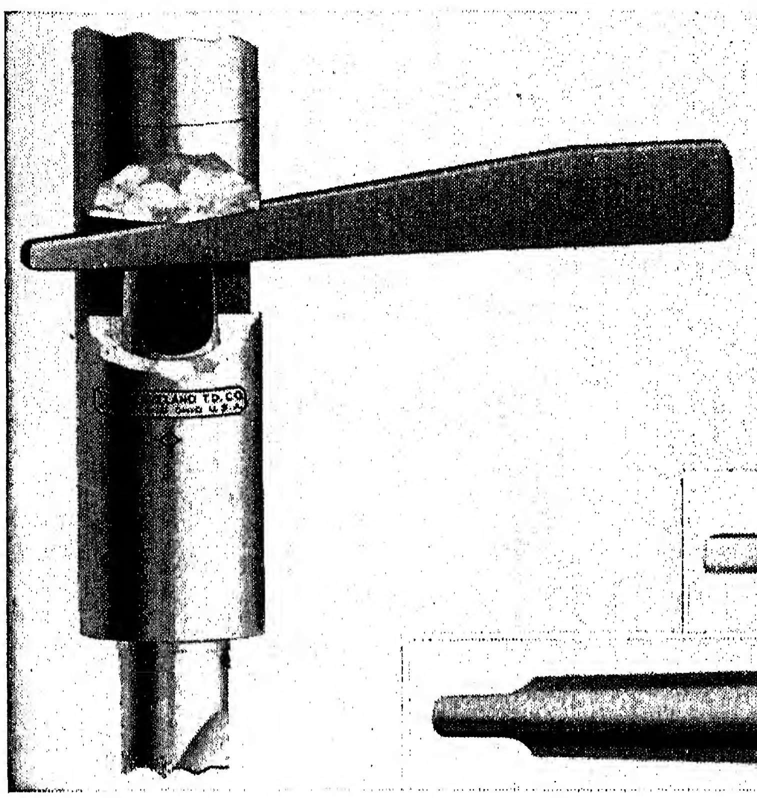
of reamer offering a compromise has alternate blades on right- and left-hand helices.

Reamer speeds and feeds. As a rule, reaming must be done at low speeds and high feeds for best results. Speeds of 50 to 75 per cent of drilling speeds and feeds 200 to 300 per cent of drilling feeds for rose reamers and 300 to 500 per cent for fluted reamers are recommended. Up to 1/16 in. of stock may be removed in rough reaming, but best results are obtained in finishing by limiting the stock to 0.004 to 0.012 in. on the diameter. Less is taken from small holes than from large ones. Too much stock impairs finish and tends to deflect the reamer blades outward, causing the reamer to cut oversize. Cast iron is sometimes reamed dry, but a cutting fluid is always desirable for reaming other metals to maintain size and get a good finish.

Taper reamers. Taper reamers are used to finish tapered holes. They may have straight or spiral flutes but generally are solid. Some are made for special tapers, but commercial reamers are available for standard tapers. The dimensions of common standard tapers are given in a later section in this chapter. Taper reamers for

standard sockets with Morse, Jarno, and Brown and Sharp tapers come in sets, with one roughing and one finishing reamer in a set. The roughing reamers have grooves or nicks along the cutting edges to break up the chips. *Taper pin reamers,* like the one in Fig. 6-19, are common for reaming holes for self-locking taper pins. A *center reamer* has 60° or 82° included angle.

Sockets. Drills, reamers, arbors, etc. often have tapered shanks of different sizes. *Taper sleeves,* like that of Fig. 6-20 A, are commonly used to adapt tools and centers to fit spindle holes. Another form is the *fitted socket* of Fig. 6-20 B that provides an extension. In addition to the tapers, a socket has a slot for the tang of a tool inserted in it and has a tang itself. The slot is open on



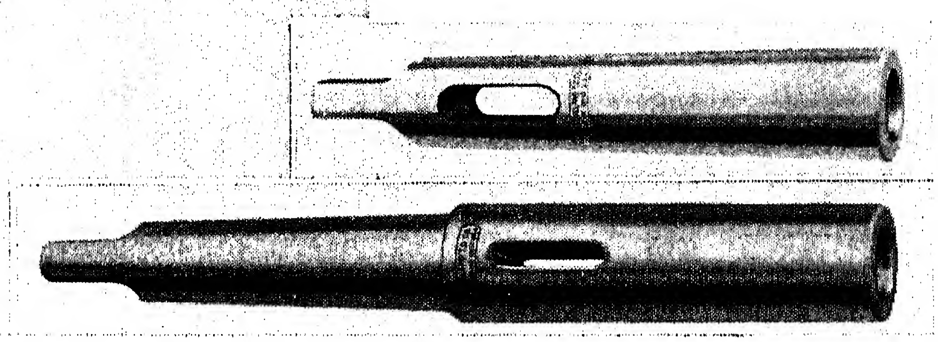


Fig. 6-20. Taper shank sockets. A. A sleeve or shell socket. B. A fitted socket. C. Use of a drift to remove a tapered shank. (Courtesy Cleveland Twist Drill Co).

the sides so that a tapered drift can be inserted, like in Fig. 6-20 C, to separate the tool from the socket.

Internal Operations on the Lathe

Procedures. A workpiece is faced before being drilled or bored to provide a true surface for starting the hole. This *facing* is like that done on the end of a shaft held between centers and previously described. A right-hand facing tool or turning tool with a nose radius may be used. If there is much stock on the end of the workpiece, one or more roughing cuts may be taken before the finishing

cut. The tool is fed toward the center for roughing and either in or out for finishing.

A long drill can be deflected easily when it starts into a workpiece. To aid in starting a drill, a center hole is first spotted in the end of the work by a center drill held in a drill chuck, like in Fig. 6-3, or by a short starting drill. Sometimes the hole is started by the full length drill for completing the hole, but then the butt end of a lathe toolholder fastened in the toolpost is brought to bear against the side of the drill to guide and support it.

When a cored hole is drilled in a casting with a four flute core drill, the drill tends to follow the rough hole and run out. A countersink or short counterbore may be put in the hole by a single point tool from the carriage of the lathe to start the drill right. This may also be done to start a rose reamer.

Drills and reamers are held by the tailstock of a lathe when the work is chucked. A straight shank drill may be gripped by a drill chuck with a tapered shank inserted in the tailstock spindle. A reamer is held in that way in Fig. 6-22. Tapered shank drills may be inserted in the spindle hole, with a sleeve if necessary. The tailstock is slid towards the headstock until the drill almost touches the work. Then the tailstock is clamped to the bed, and the drill is fed into the revolving workpiece by turning the handwheel on the tailstock. Cutting oil should be applied when drilling steel. If the hole is deep in proportion to its diameter, the drill should be withdrawn from time to time to free the chips.

A lathe tailstock spindle has no cross slot like the one in a drill press spindle to engage the tang of a drill. A large drill is subject to considerable torque when cutting and may turn instead of sticking in the tailstock spindle hole. This wears and scores the taper in the spindle. One way to prevent that is to clamp a lathe dog on the drill and let the tail of the dog bear against the top of the compound rest. Another practice is to use a drill holder as indicated in Fig. 6-21. The drill holder has a tapered hole for the drill shank

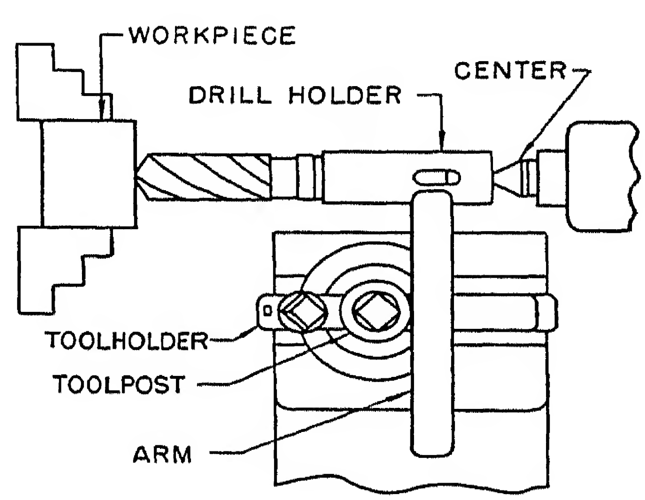


Fig. 6-21. Use of a drill holder.

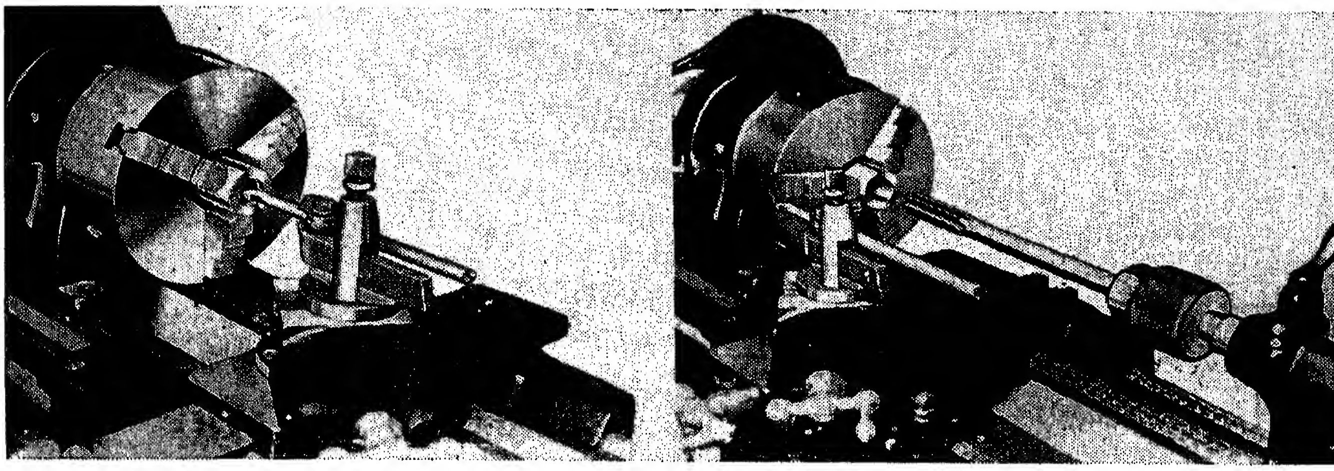


Fig. 6-22. Boring and reaming a hole on a lathe. (Courtesy South Bend Lathe Works.)

and a slot for the tang of the drill. The arm of the drill holder rests upon the shank of a toolholder and against the toolpost to counteract the torque of drilling and keep the drill from jumping ahead. The tailstock center fits in a center hole in the end of the driver and supports and pushes the holder along as the tailstock spindle is fed out. As the drill is fed into the work, the carriage is pushed along by the arm of the holder. The operator retards the carriage by means of the handwheel on the apron so that the drill and holder cannot go ahead too fast and slip off the tailstock center.

A hole is enlarged by boring to make it concentric with the axis of rotation of the workpiece. A single point boring tool is most commonly used on a lathe and is carried on the carriage as shown in Fig. 6-22. Light boring bars must be used for small holes. They spring easily and are not capable of heavy cuts.

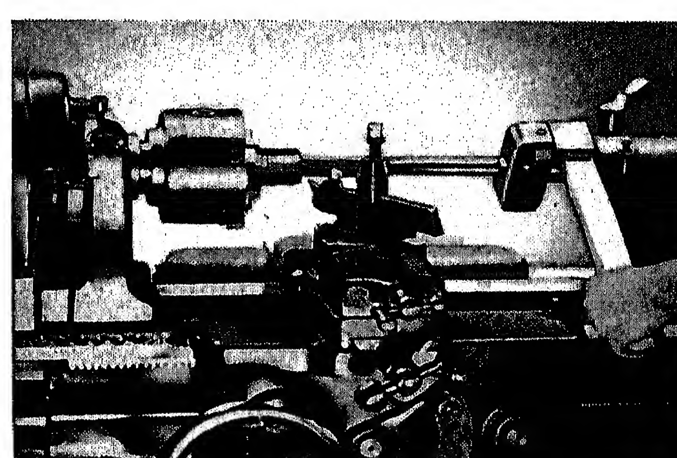


Fig. 6-23. A reamer held in a floating reamer driver. (Courtesy South Bend Lathe Works.)

A bored hole can be measured with an inside caliper. One leg is placed on one side of the hole and the other leg is made to touch a diametrically opposite spot. In that position, the caliper is adjusted so it just touches the surfaces but is not forced. Then the distance across the tips of the legs is measured with an outside micrometer cal-

iper. A finished hole of standard size is generally checked with a plug gage.

Straight shank reamers can be held in a drill chuck as in Fig. 6-22, and tapered shank reamers may be inserted in the tailstock spindle. Where it is desirable for a reamer to follow a hole bored true, a floating reamer driver similar to the one in Fig. 6-23 is used. A reamer may be supported on the tailstock center point with a dog clamped to the reamer to keep it from turning. A holder like the one illustrated in Fig. 6-21 may also be used. These allow a reamer to follow a bored hole and reduce the chance of cutting a hole with a bell mouth.

Considerations in the selection of internal operations. Holes can be machined in a number of ways. Some are just drilled; others are drilled and then bored. Some holes are drilled and then reamed; others are drilled, bored, and reamed. Boring may be done in one pass or by two or more cuts. A hole may be rough reamed with a rose reamer and then finished with a fluted reamer. Another may be bored to within 0.010 in. of size and then finished with a fluted reamer alone. Each method of machining holes has certain advantages and disadvantages that make it desirable in some cases and undesirable in others. An engineer needs to understand what each method is, the proper place for each method, and the reasons for it.

The three main considerations in the selection of a method for machining a hole are (1) the length of time required to do the work, (2) the cost or availability of the tools, and (3) the finish and accuracy required.

The time to machine a hole depends upon how fast the tool or tools can remove material and the number of cuts that must be taken. Usually, the more cutting edges on a tool, the faster it is able to remove metal. A multiblade boring tool can be expected to cut faster than a single point tool. If two tools are used, such as a drill and boring tool, the machining time is normally more than if the stock were removed in one cut by one tool alone. Conditions determine the least number of cuts that may be taken to machine a hole, but in any case it is desirable to take the fewest number of cuts possible under the circumstances.

Although multiedge tools are fast, their initial cost and upkeep are higher than that of single-edge tools. Also, drills, boring tools, and reamers with two or more cutting edges either are made in

definite sizes or can be adjustd only within limited ranges. For instance, each twist drill produces a hole of a certain nominal diameter. A boring tool with several blades may be solid and therefore capable of cutting only one diameter, or it may be adjustable over a small range. In contrast, a single point boring tool carried on the carriage of a lathe can be adjusted for any diameter within the range of the machine. A large stock of fixed size tools must be available to machine a variety of holes of many diameters. If only a few holes of each size are wanted, single purpose tools are often not economical. For such cases, single point tools are frequently used even though more time is taken to set them for each cut and for cutting. On the other hand, for quantity production of a few sizes of holes, tools for the specific sizes are usually most economical.

Drills over 1 or 2 in. diameter are not efficient, and common practice is to bore after drilling a lead hole for large holes. Even if a large drill is used, less thrust and power are required if a smaller lead hole is drilled first. The lead hole need not be much larger than the web thickness of the large drill.

Although several cuts consume more time than one, several often are necessary to remove the material from a hole and in the end produce the required finish and accuracy. A good finish is not obtained when a heavy cut is taken. Typical ranges of surface finish resulting from drilling, boring, and reaming are indicated in Fig. 2-6. Drills must be used to open holes and can remove stock rapidly but leave relatively rough surfaces. Boring tools also give rough surfaces when heavy cuts are taken but are capable of producing good finishes with light cuts, fine feeds, and high speeds. Rose reamers are designed for fairly heavy cuts and give only fair finishes. Good finishes can be obtained by using fluted reamers properly for final light cuts. Those are the methods commonly used on lathes. Other methods are available for finishing holes and often are used for quantity production. They include broaching, grinding, and burnishing for soft materials, and grinding, lapping, and honing for hard materials. Those methods will be described in later chapters.

A drilled hole cannot be held closely to a specified diameter. Under average conditions, it may be said that practical drilling tolerances range from 0.002 in. for a 3/16 in. diameter to 0.010 in. for 1 to 2 in. diameters. What is more, a drill often does not start true and frequently runs out in its course through the material.

Holes drilled in a lathe therefore can be expected to be appreciably eccentric and crooked.

Holes are bored to remove material rapidly, especially in larger sizes, and to make them straight and true. A multiblade boring tool will produce a large number of holes within 0.001 or 0.002 in. of a specified size. A single point boring tool can be arranged to cut a size within any reasonable tolerance, even within 0.001 in., but on a lathe that is a time consuming procedure that must be repeated for each workpiece. If a reamer is available, it offers a quicker way of bringing a hole close to an exact size. A fluted reamer properly used will produce a large number of holes within 0.001 in. of its nominal diameter in one quick pass for each hole.

A reamer tends to follow an original hole and cannot be depended upon to make a hole concentric or parallel to the axis of rotation of a workpiece by itself. A rose reamer will produce a straight hole after it has been given a true start, but the direction and position of the hole depends upon how the reamer is started. A hole may be drilled and then reamed for size and finish if its position and direction are not important. On the other hand, if a hole must have a definite location as well as a close tolerance on its diameter and a good surface finish, it must be bored before being reamed.

Taper Turning and Boring

Kinds of tapers. A tapered piece increases or decreases in size at a uniform rate along its length. Pieces of various shapes can be said to be tapered. The drift of Fig. 6-20 C is tapered. Some flat and square keys for shafts and hubs are tapered. However, the objects usually called tapered have round sections, like lathe centers and drill shanks. They are conical in form. Inside as well as outside surfaces may be tapered. An external tapered surface commonly has a counterpart in and is fitted to an internal tapered surface, and vice versa. For example, the tapered shank of a drill matches the inside taper of a mating sleeve.

The amount of taper may be measured by the included angle between the sides or by the angle that one side makes with the axis of a workpiece. However, the most common way of expressing taper is by the increase in diameter of a piece in a certain unit of length.

This usually is expressed in inches per foot. Thus if a piece is 1 ft long and measures 2 in. in diameter at its large end and 1½ in. in diameter at its small end, the difference in diameter is ½ in., and the amount of taper is said to be ½ in. per ft.

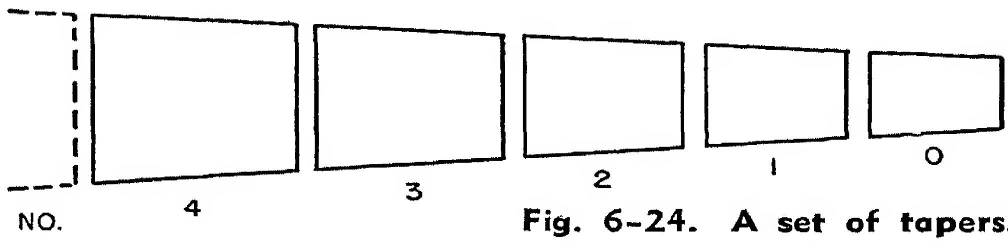


Fig. 6-24. A set of tapers.

Certain tapers have been adopted as standard to make tools and other articles interchangeable. One of these is the *Morse taper* that is approximately ⅝ in. per ft. Drilling machines and many lathes have Morse taper spindle holes. These are of various sizes designated by whole numbers 0 through 7. The sizes are in series, each one a larger section of a cone as indicated by Fig. 6-24. No. 0 is the smallest and has a diameter of 0.356 in. at the large end and a nominal length of 2 in. Number 7 has a diameter of 3.270 in. at its large end and a length of 10 in.

Other tapers used on lathes are the Reed and Jarno. The *Reed* taper is 6/10 in. per ft. In the *Jarno* series, each size number designates the length in half inches, the small end diameter in tenths of an inch, and the large end diameter in eighths of an inch. Thus a No. 6 Jarno taper has a length of 3 in., a diameter of 0.600 in. at the small end, and a diameter of 0.750 in. at the large end. The increase in diameter is 0.150 in. in the length of 3 in., or 0.600 in. in 1 ft.

The *Brown and Sharpe* taper is used on the shanks of milling cutters and milling machine attachments and accessories. Its taper is ½ in. per ft, and the sizes are designated by numbers. The taper in the main spindles of modern milling machines is the *National Machine Tool Builders Standard* taper of 3½ in. per ft, arranged in overlapping sizes and numbered by tens from 10 to 60. This taper is also called the *American Standard Steep Machine* taper.

The standard taper for locking taper pins is ¼ in. per ft. The standard pipe thread taper is ¾ in. per ft.

Taper machining methods. Tapers are turned on ordinary lathes by three methods. These involve using the compound rest, setting over the tailstock, and using the taper attachment. The method selected for an operation depends upon the length and

angle of the taper and the number of pieces to be produced. Tapers may be bored by using the compound rest or the taper attachment. Tapered holes are often bored and then finished by means of tapered reamers.

Some lathes have change gears to provide various rates of cross-feed with respect to the lengthwise feed. Both the saddle and cross-slide are fed at the same time, and the tool is moved in an angular path to cut a taper depending upon the rates of feed selected for the two units.

The cutting edge of a tool must be set at exactly the same height as the axis of the workpiece to turn or bore a true taper. That height may be gaged by the point of a lathe center.

A taper may be measured by checking two diameters with a micrometer, but accurate tapers are usually machined to fit a gage. An outside taper is tested with a tapered ring gage. A chalk mark is made along the entire length of the workpiece taper. The piece is placed in the ring gage and turned carefully. The chalk will be rubbed where the work and gage bear together. If the fit is perfect, a bearing will show along the entire length of the taper. If the bearing is only partial, adjustments are made to correct the taper. An inside taper is tested with a tapered plug gage in a similar manner.

Use of the compound rest. The compound rest of a lathe has a circular base graduated in degrees and may be swiveled and clamped at any desired angle. The rest can then be moved along a path at that angle by turning the compound rest feedscrew by hand. In this way, a cutting tool mounted on the rest is traversed across a revolving workpiece to machine a taper. Such a setup for machining both an external taper on a punch and an internal taper on a die is shown in Fig. 6-25.

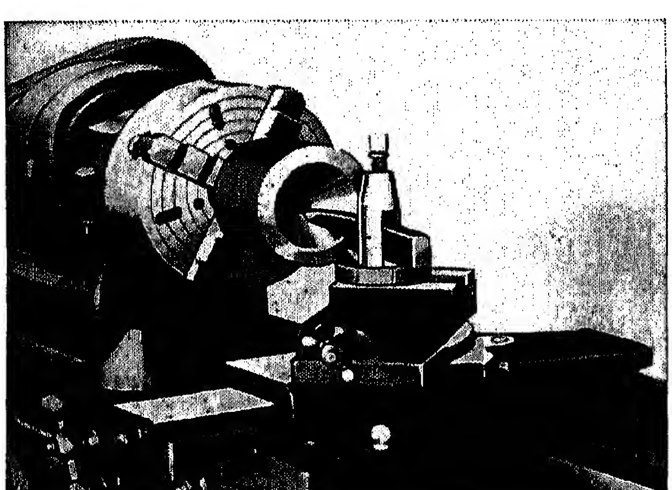


Fig. 6-25. A set up for machining a conical punch and die with the compound rest set at an angle. (Courtesy South Bend Lathe Works.)

The compound rest method can be used for steep as well as gradual tapers. However, the linear movement of the com-

pound rest slide is limited, and the method is suited only for short tapers.

Tailstock set-over. Tapers can be machined on work that can be turned between centers by setting over the top of the tailstock. That means that the tailstock center, which is in line with the headstock center for straight turning, is moved horizontally out of line. That arrangement is indicated in Fig. 6-26. The amount of set-over depends upon the over-all length of the workpiece and the degree of taper desired.

The distance in inches that the tailstock must be set over is equal to one-half of the product of the length of the workpiece in feet times the desired taper in inches per foot. This is true whether the taper is to be machined over the entire length of the piece or only over part of the length, because the position of the workpiece depends upon its length for a certain set-over, as shown in Fig. 6-26.

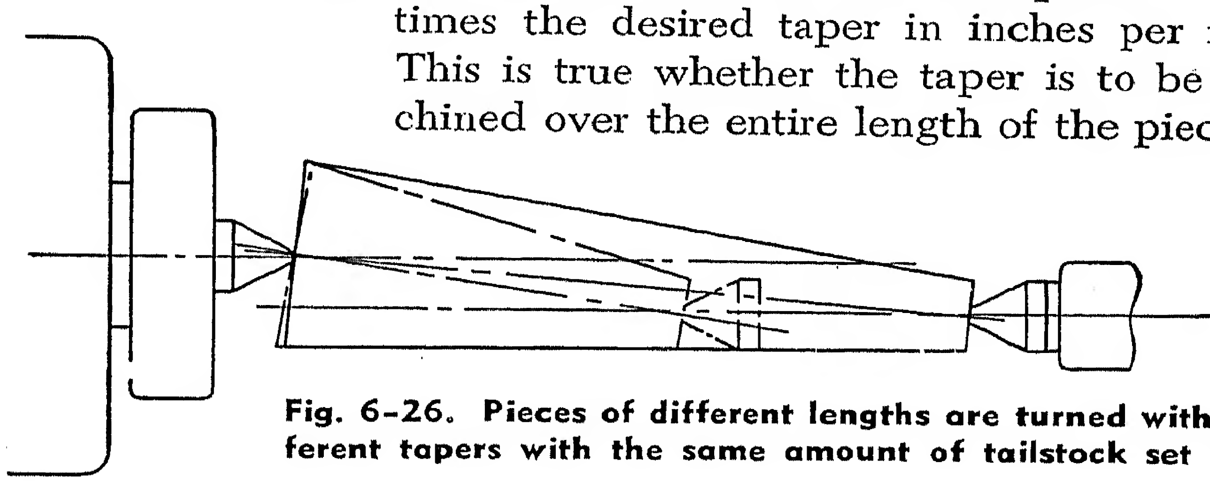


Fig. 6-26. Pieces of different lengths are turned with different tapers with the same amount of tailstock set over.

As an example, a piece 6 in. long is to have a Brown and Sharpe taper of ½ in. per ft. turned on it. The amount of set-over required is

$$\frac{1}{2} \times \frac{6}{12} \times \frac{1}{2} = \frac{1}{8} \text{ in.}$$

The diameters at the ends of a taper and the lengths of the taper and workpiece in inches may be specified instead of the taper in inches per foot. If the length of the taper is the same as the length of the workpiece, the amount of set-over must be one-half the difference between the large and small

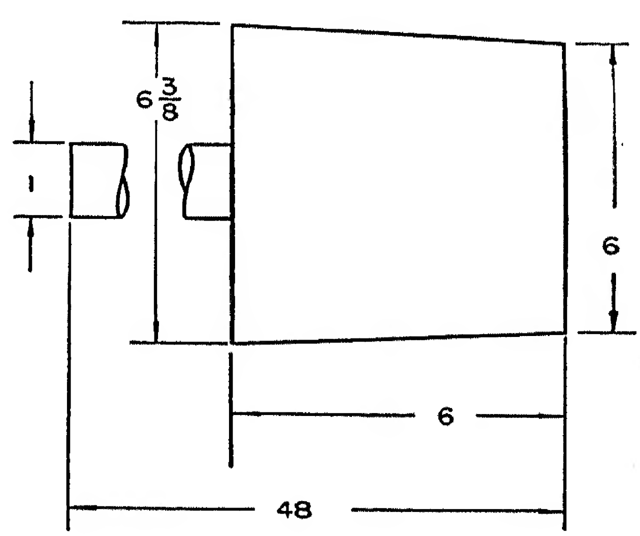


Fig. 6-27. A workpiece with a taper.

diameters. If the taper is shorter than the over-all length of the workpiece, the quotient of the length of the workpiece divided by the length of the taper is multiplied by one-half the difference between the large and small taper diameters to find the amount of set-over in inches. An example is given by Fig. 6-27. The set-over required for that piece is

$$\frac{48}{6} \times \frac{6\frac{3}{8} - 6}{2} = 8 \times \frac{3}{16} = 1\tfrac{1}{2} \text{ in.}$$

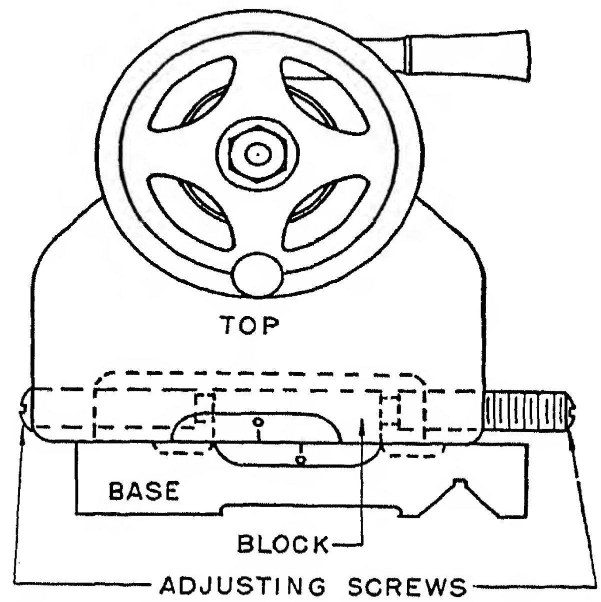


Fig. 6-28. A lathe tailstock set over for taper turning.

An end view of a typical tailstock in Fig. 6-28 shows the means for set-over adjustment. The bolt that clamps the tailstock to the bed of the lathe also clamps the top to the base of the tailstock. The nut on the clamping bolt is first loosened. Two adjusting screws in the top bear against a block projecting from the base. One adjusting screw is loosened, and the other tightened to set over the top. A line on the top and one on the base coincide when the top is on center for straight turning. These lines are moved apart when the top is set over. On some lathes a scale is provided opposite one of the lines to indicate the displacement. The tailstock is clamped to the bed to secure the setting.

To turn an accurate taper, a trial cut is made after the tailstock has been set over. The tapered surface is checked with a gage. The tailstock is then readjusted to correct for mis-taper, another trial cut is taken, and so on until the desired accuracy is obtained.

Taper attachment. The taper attachment described in Chapter 5 and illustrated in Fig. 5-19 is used for turning and boring tapers. The taper attachment is easy to adjust and eliminates the necessity of setting over the tailstock for turning. It may be set permanently for a specific taper and disconnected to allow straight turning between times. Long tapers can be machined with the taper attachment.

The taper attachment illustrated in Fig. 5-19 has an upper and lower slide bar. The lower slide bar is supported and slides on an

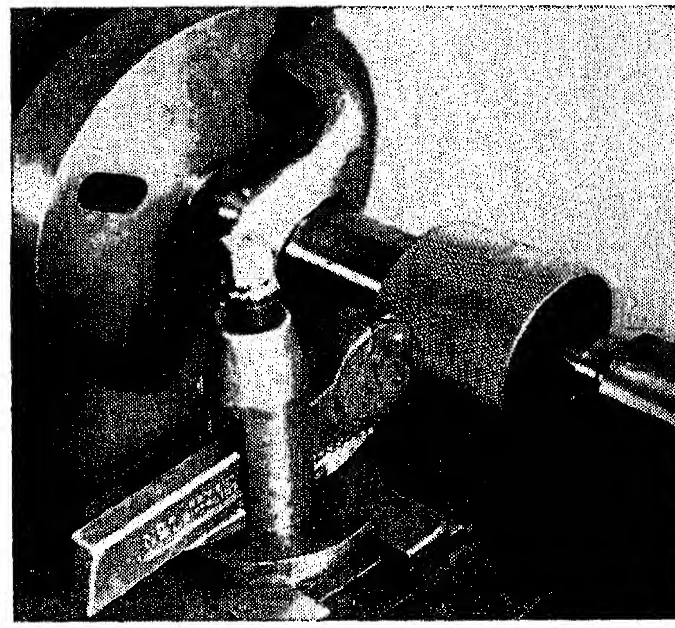


Fig. 6-29. Knurling a piece in a lathe. (Courtesy South Bend Lathe Works.)

extension on the rear of the saddle. It is attached to a bracket that is clamped to the bed to secure the slide in a position alongside the taper to be machined. The upper slide can be swiveled on the lower slide. It is set to the desired taper by a scale on its end and clamped at the setting. Taper attachments are commonly graduated in inches per foot of taper and degrees. The tool is adjusted to the work, and a binding screw tightened to fasten an extension of the cross-slide to a block that moves along the upper slide. If the cross-feed screw is not telescopic, it must be disconnected from the cross-slide. Then as the carriage is traversed lengthwise on the bed, the cross-slide is moved in or out uniformly by the block which follows the upper slide. The tool is thus guided in a path parallel to the upper slide of the attachment. A trial cut is taken, and the taper is checked. Usually the attachment must be readjusted because the upper slide cannot be set exactly the first time from the graduations on its end alone. Before the final finishing cut is taken, successive cuts must be taken and adjustments made until the taper is correct.

Miscellaneous Operations

Knurling. Knurling is the process of embossing a diamond-shaped pattern on the surface of a workpiece. A special tool with serrated rollers is held in the toolpost and pressed against the revolving workpiece as shown in Fig. 6-29. The patterns of all knurled surfaces are similar, but the impressions differ in size and depth for fine, medium, and coarse knurls.

Knurling is done at the slowest speed available on a lathe, and plenty of oil is flowed on the tool and workpiece. The tool is applied at the right-hand end of the surface until the knurl is about 1/64 in.

deep. Then the tool is given a coarse feed along the surface of the work by power. Both knurling rolls must bear on the work, and care must be taken that they track to form a clean, sharp pattern.

Filing, polishing, and lapping. Turned surfaces can be made smooth and bright by filing and polishing. A fine mill file is stroked slowly across the revolving work surface. The workpiece should turn at a rate of two or three revolutions for each stroke of the file. Only enough stock is removed to take off the tool marks. Too much filing can make the work surface uneven and inaccurate. Polishing with successively finer grades of emery cloth after filing results in a very smooth, bright surface. The lathe is run at high speeds, and oil is used on the emery cloth.

Internal hardened surfaces are often cleaned and given smooth finishes by lapping on a lathe. Coated abrasive cloth or abrasive particles are commonly used with oil. A rod or mandrel of the right size is held and revolved in a chuck. The abrasive and oil are applied to the lap, and the workpiece, held by hand, is slipped over it.

Pieces generally are polished or lapped on a lathe in small quantities only. In large quantities, they are ground or finished by methods described in Chapters 17, 18, and 19.

Cutting off. Cutting off or *parting* is the operation of severing a workpiece from a bar. It is done with a narrow tool as indicated in Fig. 4-10. The blade for parting is as narrow as the material and depth of cut will allow. The blade has clearance on both sides and is tapered back from the cutting edge. Consequently, a cut-off tool is relatively weak, and care must be taken that it is set square and on center and fed slowly so as not to be overloaded. A cut-off tool has no side rake because it cuts only on its front edge.

Questions

1. What is the difference between plain or straight turning and chuck work?
2. Describe several ways of centering workpieces.
3. Describe how a lathe is set up for straight turning.
4. How is a lathe tool normally set for straight turning?
5. Describe how a lathe is set up for chuck work.
6. In what sizes are standard drills available?

7. Sketch a two flute twist drill and name and designate its parts and angles.
8. What are the effects of incorrect drill grinding?
9. Describe common types of boring tools.
10. What are the differences between a hand and machine reamer?
11. How does a rose reamer differ from a fluted chucking reamer in features and purpose?
12. What is the difference between an expansion reamer and an adjustable reamer?
13. Sketch a machine reamer and designate its principal parts and angles.
14. How is a sticking taper tool shank removed from a socket?
15. How may drills and reamers be held on a lathe?
16. What considerations determine the method selected for machining a hole on a lathe? Discuss the methods that may be used to satisfy various conditions.
17. State the taper per foot of the Morse, Jarno, Brown and Sharpe, and taper pin tapers.
18. By what three methods may tapers be machined on a lathe? What advantages and disadvantages does each offer?

Problems

1. A taper 6 in. long has a large diameter of 2.0625 in. and a small diameter of 1.500 in. What should be the setting of the taper attach ment in inches per foot to machine this taper?
2. A workpiece is to have a taper turned over its full length. Its large diameter is 2.0625 in., its small diameter 1.625 in. How much must the tailstock be set over?
3. A taper is 3.9375 in. long and has a large diameter of 1.8125 in. and a small diameter of 1.0313 in. What is the taper in inches per foot?
4. The small diameter of a taper is 0.375 in. Find the large diameter if the taper is 2.500 in. per foot and the length of the taper is 3.8125 in.
5. The large diameter of a piece measures 0.9375 in., the small diameter 0.4375 in. The taper is 4 in. per foot. What is the length of the taper?
6. The over-all length of a piece is 16 in., and its taper is 3 in. per foot. What must be the tailstock set-over?

7. Calculate for the workpiece of Fig. 6–30 A:
   (a) the taper per foot; (b) the tailstock set-over.
8. Calculate for the workpiece of Fig. 6–30 B:
   (a) the taper per foot; (b) the tailstock set-over.
9. Calculate for the workpiece of Fig. 6–30 C:
   (a) the taper per foot; (b) the tailstock set-over; (c) the angle in degrees to which the compound rest should be swiveled.

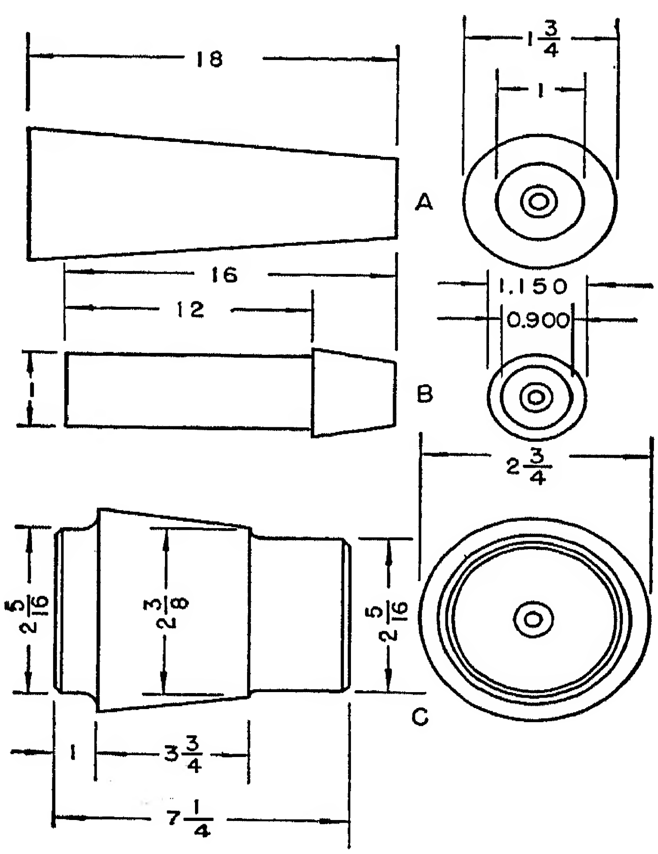


Fig. 6–30. Tapered pieces.

References

How to Run a Lathe. South Bend, Ind.: South Bend Lathe Works, 1947.

Woodstock, Frederic L., *Design of Metal Cutting Tools.* New York: McGraw-Hill Book Co., 1948.

Manufacturers' Catalogs:
Chicago-Latrobe Twist Drill Works, Chicago 10, Ill.
Cleveland Twist Drill Co., The, Cleveland, Ohio.
Eclipse Counterbore Co., Detroit, Mich.
Standard Tool Co., The, Cleveland, Ohio.

Chapter 7

SCREW THREADS AND SCREW CUTTING

Screw Threads and Screws

Nature of screw threads. A *screw thread* is a ridge of uniform section that lies in a helical or spiral path on the outside or inside of a cylinder or cone. The groove between the ridges is called the *space.* A *straight thread* lies on a cylinder; a *tapered thread* lies on a cone. A thread on an outside surface is an *external thread.* A screw has an external thread. An inside thread is an *internal thread* and is found in a nut.

A *right-hand thread* is one that turns clockwise as it moves away from the observer. A *left-hand thread* turns counterclockwise from the same position. A thread is understood to be right-handed unless designated otherwise, and most screw threads are of that hand. Sometimes left-hand threads are convenient, such as on the cross screw of a lathe or on the left-hand end of an axle.

Uses of screw threads. Screw threads are almost indispensable for many purposes. They act as fasteners, transmit power and motion, and serve as measuring devices. Machine screws, bolts, studs, and nuts are universally used to fasten together the parts of most mechanical devices. They hold securely, yet can be removed easily without damage to the parts. Screw jacks transmit power and increase forces. Leadscrews of lathes and other machine tools make controlled, precise, and uniform movements possible. Accurate screws in micrometers, calipers, dividing machines, etc. magnify movements so that fine measurements can be made easily.

Features of a screw thread. The chief features of an external thread are illustrated by the section of Fig. 7-1. Internal threads

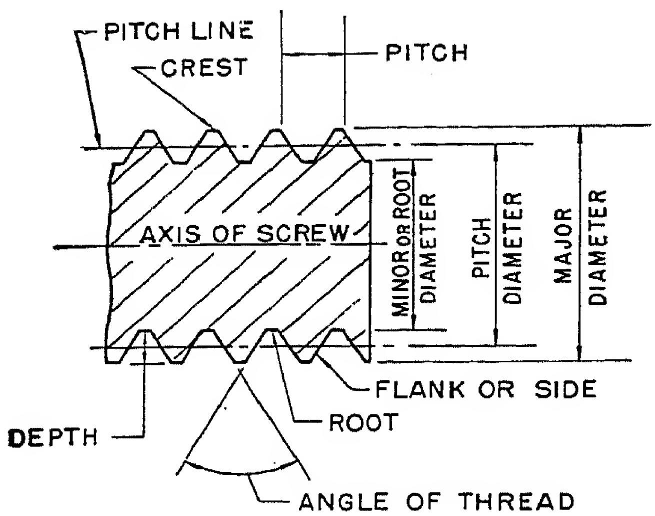


Fig. 7-1. Parts of a screw thread.

have corresponding features. They determine the size and shape of a thread. The important ones are covered here.

The *pitch* is the distance parallel to the axis from any point on a screw thread to a corresponding point on the next ridge. The pitch is the reciprocal of the *number of threads* in a unit of length, usually an inch. Thus, a screw that has eight threads per inch has a pitch of ⅛ in.

The *lead* is the distance a screw advances axially in one full turn. The lead is the reciprocal of the *number of turns* required to advance the screw axially an inch. Thus, a screw that requires 8 turns to move forward an inch in a nut has a lead of ⅛ in.

A *single thread screw* has only one continuous thread on its surface. Most commercial screws, bolts, and nuts have single threads. A *multiple thread screw* has two or more separate threads. A *double thread screw* has two threads, a *triple thread screw* has three, etc. The lead of a single thread screw is equal to its pitch, but the lead of a double thread screw is twice the pitch, the lead of a triple thread screw is three times the pitch, etc.

A double thread screw advances twice as far in one turn as does a single thread screw of the same pitch. The more threads in a screw, the faster it advances as it turns. Thus, multiple thread screws are useful for transmitting motion rapidly. Also, the more threads, the finer the pitch and the shallower the space between threads for a specific lead. That makes the screw stronger. A fountain pen screw cap usually has multiple threads. It can be put on with a few turns, and the cap and pen barrel are not grooved deeply and weakened.

The *root* is the bottom of the space between threads, and the *crest* is the top of a thread. The *flank* is the surface on the side of a thread between the crest and root. The *angle* of a thread is measured between two flanks. The *depth* of a space is the same as the *height* of a thread, both measured radially.

The *major diameter* of a straight thread is the diameter of a

cylinder in which the crest of an external thread or the root of an internal thread would lie.

The *minor diameter* of a straight thread is the diameter of a cylinder in which the root of an external thread or the crest of an internal thread would lie.

The *pitch diameter* of a straight thread is the diameter of an imaginary coaxial cylinder that cuts the thread where the width of the thread is equal to the width of the space.

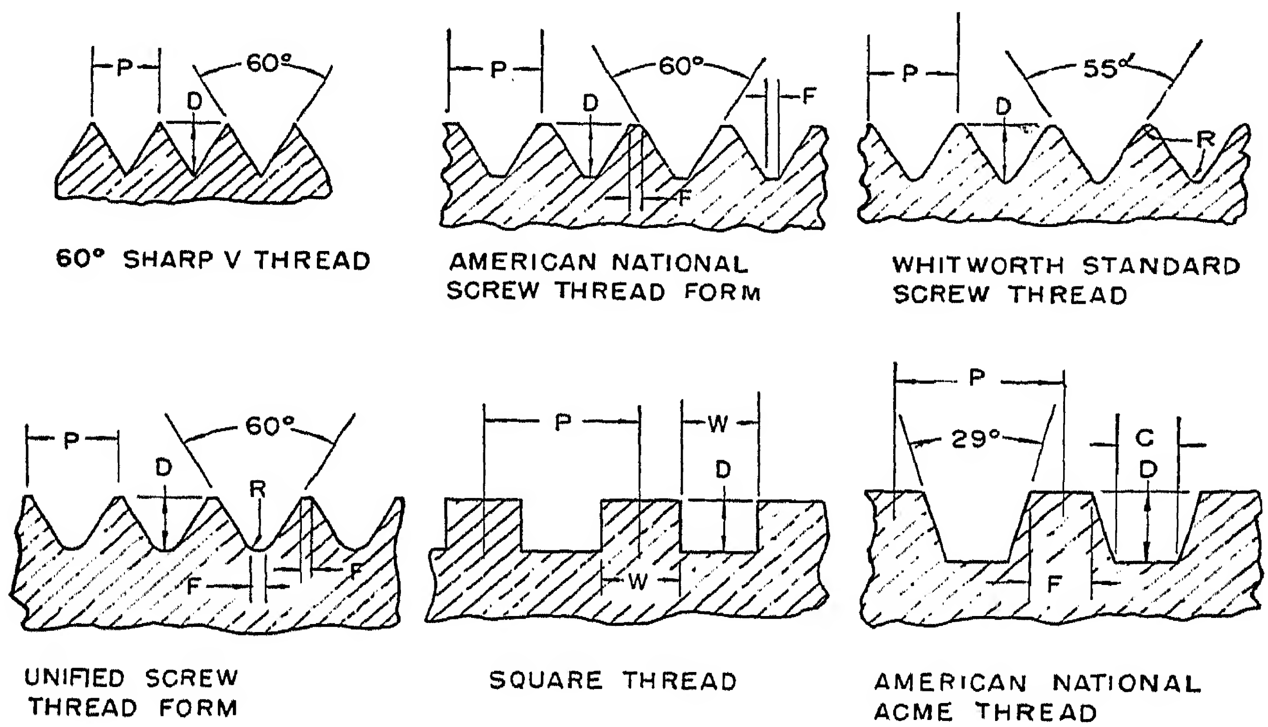


Fig. 7-2. Cross-sections of various forms of screw threads.

Screw thread forms. Screw threads are made with a number of cross-sections. The most common are indicated in Fig. 7-2.

The *sharp V thread* has an angle of 60°, and each flank makes an angle of 30° with a radius of the screw. The thread and space have the same form. The depth (D) = 0.866 × pitch (P).

The *American National Screw Thread Form* is a modification of the V thread with flat crests and roots. The depth (D) = 0.6495 × pitch (P). Thus, this thread has only ¾ of the depth of a sharp V thread. The width of flats (F) = P/8. The angle is 60°. This is the most common thread form for machine screws and bolts in the United States.

The *Whitworth or British Standard Screw Thread* has an angle of 55°. The crests and roots are rounded with a radius (R) = 0.1373 ×

pitch (P). The depth (D) = 0.6403 × pitch (P). This form is seldom used in the United States.

The *Unified Screw Thread Form* is substantially the same as the American National Screw Thread Form with an included angle of 60°. Rounding is permitted in the root and on the crest but must be kept within the limits of the major and minor diameters.

The *Square Thread* has a width (W) and depth (D) equal to half the pitch (P). The thread sides are parallel. When a Square thread is cut, the tool for either the screw or nut, but not both, is made 0.001 to 0.003 in. wider than half the pitch to provide a working clearance between the mating threads.

The *American National Acme Thread* has a 29° angle. The basic depth (D) is half the pitch (P). The width of the crest (F) = 0.3707 × pitch (P), and the width at the root (C) is 0.003 to 0.005 in. less.

Square and Acme threads are able to carry heavy loads. Typical uses are for jack screws and feed and operating screws of machine tools. The Acme thread is easier to machine with uniform accuracy than the Square thread.

The *Worm Thread* has the same 29° angle as the Acme thread but a deeper space and smaller flats. It is used mostly for driving gears.

The *Buttress Thread* has one flank square or at a large angle with the axis of the screw. The other flank is inclined at 45° with the axis. The thread cross-section resembles a tooth of a ratchet, and the thread is sometimes called a ratchet thread. It is capable of taking a large thrust in one direction only.

Metric Threads resemble the National Screw Thread Form but are dimensioned in the Metric System, in millimeters instead of inches.

Threads developed on cones or tapers have special applications, such as for pipe threads. The most common of these is the *American National Taper Pipe Thread* that has an angle of 60° between flanks of the thread. The crests and roots are cut to follow a taper of ¾ in. per foot.

Screw thread standards. At one time screw threads lacked uniformity in size and shape. Each manufacturer produced screws according to his own system and standards. One product could not be interchanged with another. Standardization of sizes, shapes, and

Excerpts of American and Unified
Screw Thread Standards

Nominal Size	Basic Major Diam. in.	Coarse Thread Series (NC and UNC) TPI	Coarse Thread Series (NC and UNC) Basic Pitch Diam. in.	Fine Thread Series (NF and UNF) TPI	Fine Thread Series (NF and UNF) Basic Pitch Diam. in.
1/4	0.250	20	0.2175	28	0.2268
5/16	0.3125	18	0.2764	24	0.2854
3/8	0.375	16	0.3344	24	0.3479
7/16	0.4375	14	0.3911	20	0.4050
1/2	0.500	13	0.4500	20	0.4675
9/16	0.5625	12	0.5084	18	0.5264
5/8	0.6250	11	0.5660	18	0.5889
3/4	0.7500	10	0.6850	16	0.7094
7/8	0.8750	9	0.8028	14	0.8286
1	1.000	8	0.9188	12	0.9459
1 1/8	1.125	7	1.0322	12	1.0709
1 1/4	1.250	7	1.1572	12	1.1959
1 3/8	1.375	6	1.2667	12	1.3209
1 1/2	1.500	6	1.3917	12	1.4459

Fig. 7-3 Excerpts of American and Unified Screw Thread Standards.

pitches has been undertaken to create a condition of order. An early proposal for coarse threads was known as the United States Standard Screw Thread. Later the S.A.E. Thread System came into being in the automobile industry where fine threads often are needed.

On the basis of the two older standards, the American National Form Screw Thread Standard was adopted in 1924. It specified a series of coarse screw threads like the United States Standard Threads and a series of fine screw threads corresponding to the S.A.E. Standard. In 1935 the American Standards Association designated the symbol NC for the coarse thread series and NF for the fine thread series.

Basic dimensions and threads per inch (tpi) for some of the

standard sizes of the American National Form Screw Thread are given in Fig. 7-3. For each nominal size, the coarse thread series has fewer threads per inch than the fine thread series. In each series, the number of threads per inch decreases as the size increases. Another but less often used series in the Standard is that for extra fine threads, designated by the symbol NEF. The complete Standard also specifies the form of thread, the basic dimensions for all accepted sizes in each series, and the tolerances for various classes of threads.

An accord was reached by the United States, Britain, and Canada in 1948 on unified thread specifications. The Standard adopted is known as the Unified Screw Thread Form. The pitches, basic dimensions, and tolerances for sizes ¼ in. and larger are substantially the same for the Unified and American Threads. The coarse thread series of the Unified System is designated by UNC, the fine thread series by UNF.

Standard specifications have been set up for other thread forms, like Acme and Pipe Threads, and are given in standards bulletins and handbooks.

Classes of screw threads. Screw threads are divided into *classes* to designate the fits between internal and external mating threads. For some applications a nut may fit loosely on a screw, in other cases they must go together snugly. The different fits are obtained by assigning appropriate pitch diameter tolerances and allowances to the threads in each class. The thread form and lead are assumed theoretically correct. Actually errors do exist in those thread elements, and they leave smaller working tolerances than indicated by the standard specifications. Screws of one class may be used with nuts of another class, and that often gives desired results most economically.

The Unified Form Thread Standard recognizes several classes of threads. Classes 1A, 2A, and 3A are for screws, Classes 1B, 2B, and 3B for nuts. Classes 1A and 1B are for a loose fit, where quick assembly and rapid production are important and shake or play is not objectionable. Classes 2A and 2B provide a small amount of play to prevent galling and seizure in assembly and use, and sufficient clearance for some plating. Classes 2A and 2B are recommended for standard practice in making commercial screws, bolts, and nuts. Classes 3A and 3B have no allowance and 75 per cent

of the tolerance of Classes 2A and 2B. A screw and nut may vary from a fit having no play to one with a small amount of play. Only high-grade products are held to Class 3 specifications. A Class 4 has been included in the American National Form Standard as an interference fit. However, it requires close control of tolerances, and difficulty has been experienced in manufacturing threads interchangeably to meet the standard.

Measuring Screw Threads

The size of a screw or bolt is designated by its outside diameter. A ½ in. UNC screw should fit a nut of the same nominal size. However, this does not mean that the outside diameter of a screw can be measured alone to determine whether the screw is correct. The major and minor diameters of screw threads are dimensioned to clear the corresponding surfaces of mating threads. Threads make actual contact with each other on their flanks. Consequently measurements must be made in the space against the flanks of a thread to find its true and effective size. The dimensions that are important are the pitch diameter, thread angle, and pitch or lead.

The pitch diameter of a Unified Form or American National Form Thread equals the outside diameter minus the depth of the space. The depth of space equals 0.6495 divided by the number of threads per inch. Pitch diameters given in tables like Fig. 7-3 are derived in that way.

Screw thread micrometer caliper. A screw thread micrometer caliper has a spindle with a conical point and an anvil with two V-shaped ridges as shown in Fig. 7-4. It makes

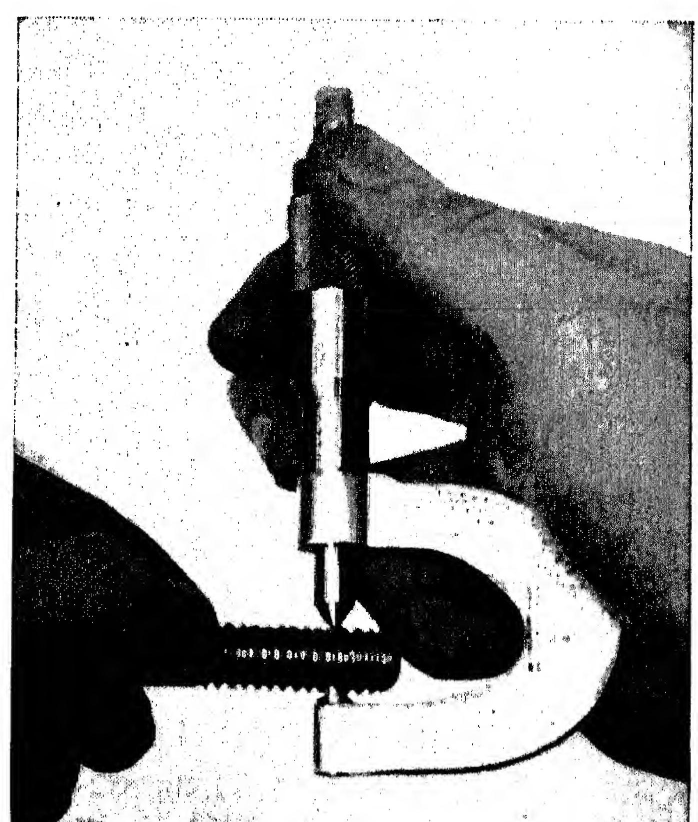


Fig. 7-4. Measuring a screw with a screw thread micrometer. (Courtesy Brown and Sharpe Mfg. Co.)

contact on the sides of a screw thread and measures the pitch diameter directly. Any one anvil is limited to a small range of pitches. Even within the proper range, the readings are slightly distorted unless the micrometer is set to a standard thread plug and used to measure threads of the same diameter and pitch as the plug.

Measuring screw threads with three wires. The three wire method of measuring pitch diameters is more accurate but slower than the use of screw thread micrometer calipers. The arrangement of the wires is indicated in Fig. 7-5. Three wires of any one diameter that would fit within the space might be used, but the preferred diameter or *best wire* for each pitch makes contact on the flanks of the thread at the pitch diameter. A best wire has a diameter equal to $\frac{2}{3}$ of the depth of a perfect V thread. That depth is equal to 0.866 divided by the number of threads per inch. Thus, the best wire diameter is equal to 0.57735 divided by the number of threads per inch.

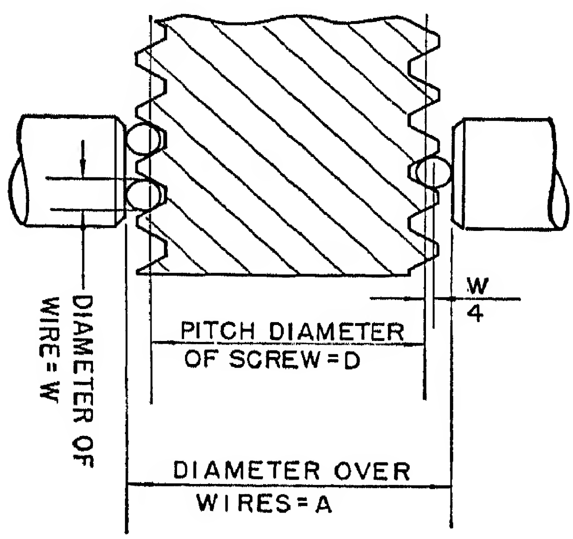


Fig. 7-5. Measuring a screw with three wires.

When a wire of diameter W touches a thread, its center lies at a distance W/4 outside the points of contact as shown in Fig. 7-5. Therefore, for a best wire size, the diameter (A) over wires is equal to the pitch diameter (D) plus the product of $\frac{3}{2}$ times the diameter of the wire (W). Tables are available that give the diameter of the best wire and the measurement over the wires for common pitches. The distances over wires are often measured by micrometer calipers, but for precise measurements, comparators, gage blocks, and optical flats may be used.

Checking pitch and thread form. If the angle or pitch of its thread is not just right, a screw acts large. A nut that fits on a perfect screw could be tight on a screw of the same pitch diameter as the perfect screw but with an incorrect angle or pitch. Thus, pitch diameter only tells part of the story. The angle and pitch of a thread must also be measured to find out how accurate a thread is.

Screw thread pitch gages illustrated in Fig. 3-32 may be used for roughly checking thread pitches.

Figure 3-35 shows an optical or projection comparator set up for checking the form of a thread. A screw is mounted on a table and positioned so that it interrupts a beam of light. The resulting shadow is magnified up to 125 times and is cast upon a screen. The image is compared with an enlarged chart and reveals clearly inaccuracies and irregularities in the thread form.

Thread gages. Ring and plug thread gages are made closely to theoretical sizes and forms of threads. They check whether screws and nuts will fit properly with their mating parts as intended. Some ring plug gages are solid, but the preferred style is adjustable like the one in the upper right-hand corner of Fig. 3-30. An adjustable ring gage is split and can be expanded or contracted a little by a locking screw and set to a master plug for size. Ring thread gages often are used in "go" and "not go" sets.

Plug thread gages are solid. A "go" and "not go" plug thread gage, like the one third from the bottom on the right of Fig. 3-30, is used to check whether an internal thread lies within certain limits. A plug thread gage for a tapered thread has only one member. If the workpiece is correct, the plug enters the hole only a certain number of turns. Such a gage has a flat on one side to show how far it should enter a tapered threaded hole that is correct.

Thread snap gages are used to check external threads rapidly. Such a gage has a frame like the snap gages in Fig. 3-30 but accurate threaded rolls in place of the anvils. Two sets of rolls provide "go" and "not go" limits. The screw that has the correct size passes the "go" threaded rolls but is stopped by the "not go" pair.

Ways of Making Screw Threads

Screws may be cut or formed. *Chasing* on a lathe with a single point tool is the most versatile method of thread cutting. External and internal, right- and left-hand, straight and tapered, and practically all sizes and pitches of threads can be chased on screw cutting

engine lathes with regular equipment. However, chasing is relatively slow and is used where only a few pieces of any one kind are to be produced. Threads with accurate leads can be cut on lathes with precision lead screws.

Multiple tooth cutters for threads include dies, taps, and thread milling cutters. Each is suitable only for certain pitches or range of sizes, but they are capable of rapid production. Dies and taps may be used by hand, on lathes, turret lathes, automatic screw machines, thread cutting machines, and drill presses. Thread milling cutters are used on thread milling machines.

Threads are formed on screws and bolts by rolling or pressing on thread rolling machines. Internal and external threads may be rolled in thin tubing. The threads on the cap of a thermos bottle or on the plug of an electric light bulb are rolled.

The methods just named for cutting and forming threads will be described in more detail in the remainder of this chapter. Threads also are ground for finish and accuracy. Grinding is the most suitable method of finishing threads in hard materials. Threads can be ground entirely from hardened stock or can be rough cut while the material is soft and finished after hardening. This process is described in Chapter 17. Other methods for producing threads include die casting, plastic molding, and hobbing.

Thread Cutting on a Lathe

Tools. A single point tool for cutting a thread on a lathe must be ground carefully to the shape of the space of the thread. Template gages are available for checking the cutting edges of tools for common thread forms. The center gage, illustrated in Fig. 7-6, is designed for gaging and setting tools for V-shaped threads. Similar gages are made for Acme and Worm threads.

Thread cutting tools are usually ground flat on top, without rake. An exception is sometimes made for V-thread tools for steel, and they are given side rake. The tools are sharpened by grinding the top face.

Sufficient front clearance must be provided so that the leading

side of a threading tool point does not drag on the side of the thread. The normal amount of clearance is usually sufficient to keep the tool from rubbing except for long leads. More clearance is required for internal than for external threading tools. Care must be taken when grinding tools for Acme, Square, or multiple threads that the side clearance angle is greater than the helix angle of the thread. The tangent of the helix angle on the periphery is equal to the lead divided by the circumference of the workpiece. The helix angle is greater at the root than at the crest of the thread.

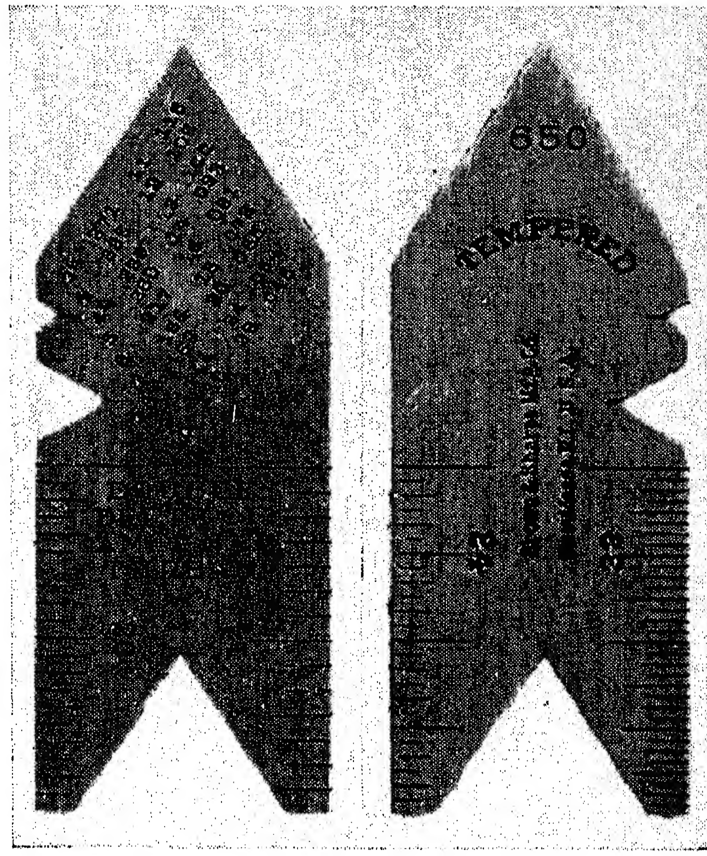


Fig. 7-6. Front and back views of a center gage.

Gearing the lathe. The number of threads per inch cut on a lathe depends upon the relative rates of rotation of the work spindle and leadscrew. Many modern lathes have a quick change gear box in the leadscrew drive, as described in Chapter 5 and illustrated in Figs. 5-1 and 5-4. A chart on the box tells how to set the drive for cutting the available numbers of threads per inch.

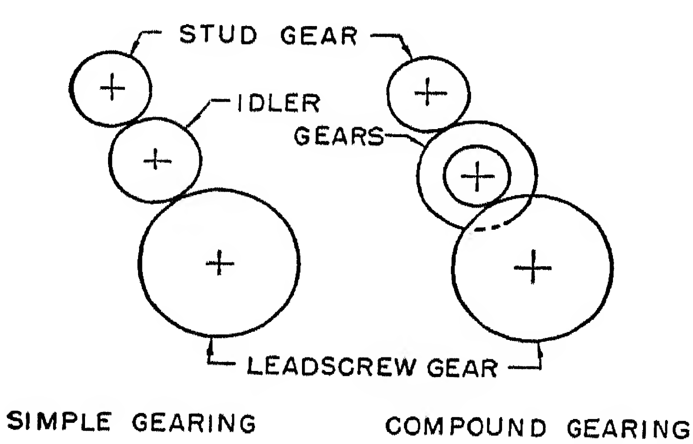


Fig. 7-7. A diagram of change gears to drive the leadscrew on a lathe.

Some lathes, particularly older ones, have pick-off change gears to drive the leadscrew, as shown in Fig.5-6. With *simple gearing*, as shown in Fig. 7-7, the rate of rotation of the leadscrew depends only upon the stud gear and leadscrew gear. The idler gear may be of any convenient

size to connect the others together. The proper gears to be applied are found from the proportion that

$$\frac{\text{the number of teeth on stud gear}}{\text{the number of teeth on leadscrew gear}} = \frac{\text{the number of threads per inch on leadscrew.}}{\text{the number of threads per inch to be cut}}$$

As an example, 13 threads per inch are to be cut, and the leadscrew has 6 threads per inch. Then:

$$\frac{\text{stud gear}}{\text{leadscrew gear}} = \frac{6T}{13T} = \frac{12T}{26T} = \frac{18T}{39T} = \frac{24T}{52T} \text{ etc.}$$

The top and bottom of the proportion may both be multiplied by the same number to arrive at convenient sizes of gears.

At times gears cannot be found for simple gearing to cut a desired thread. Then it is necessary to resort to *compound gearing.* One such arrangement is indicated in Fig. 7-7. Two idler or intermediate gears revolve together on one shaft. One idler gear is driven by the stud gears, and the other drives the leadscrew gear. The ratio of the gear train then determines the ratio between the number of threads on the leadscrew and on the workpiece.

Chasing threads. A series of light cuts is taken to chase a thread accurately and smoothly. If the cuts are too heavy, the tool may be damaged, the workpiece distorted, or the threads torn. The tool may be fed straight in for each cut on a V-type thread, as indicated in Fig. 7-8 A, for cutting cast iron or other materials that produce a discontinuous chip. For cutting steel and especially for coarse threads, the tool is fed at an angle, as shown

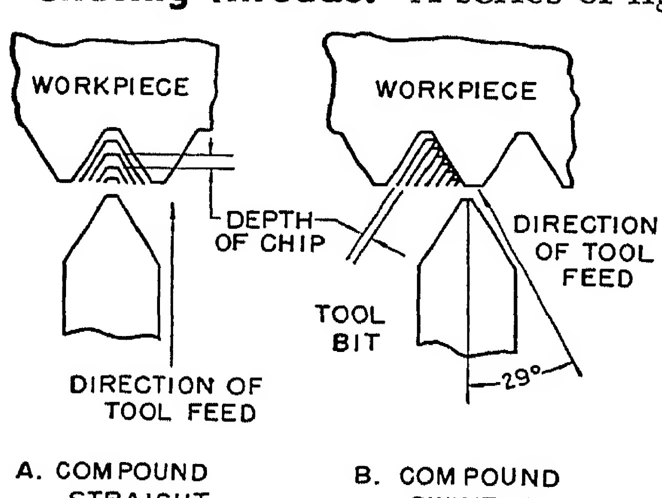


Fig. 7-8. The two ways of infeeding a threading tool.

in Fig. 7-8 B. In this way the bulk of the chip comes off only one side of the tool and can curl without interference. This makes full use of side rake if the tool is ground with it. The tool is fed straight in when cutting Square, Acme, and Worm threads.

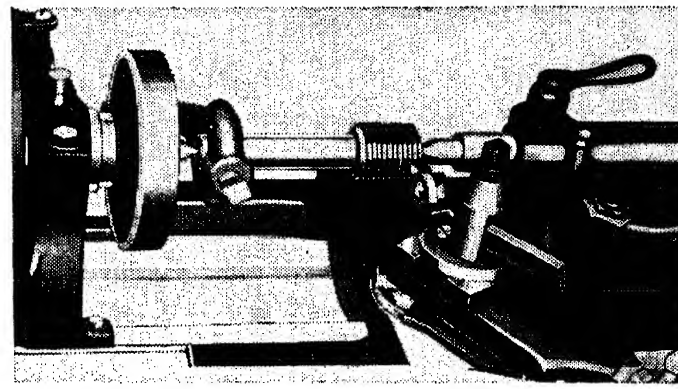


Fig. 7-9. Cutting a screw thread with the compound rest set at 29°. (Courtesy South Bend Lathe Works.)

If the tool is fed in obliquely, the compound rest is swiveled to an angle of 29°, and feeding is done by moving the compound slide. A thread cut with the compound rest set at 29° is shown in Fig. 7-9. An angle of 29° is chosen instead of the 30° half angle of a V-type thread so that the tool will clean up the side of the thread and leave a good finish.

The top of a tool should be set at the same height as the center of the workpiece for cutting a thread. The tool must also be squared with the workpiece to cut a true thread. A center gage is used to set a V-thread tool as shown in Fig. 7-10.

The tool must follow the same path each time it traverses the thread. If the lathe is equipped with a thread dial described in Chapter 5, the half-nut is engaged at the beginning of each pass when the dial position so signifies. On lathes without a feed dial, the tool is withdrawn from the cut, and the spindle is reversed to bring the carriage back to the starting position. The half-nut is left engaged until the thread is completed.

A groove is often cut where a thread ends as indicated in Fig.

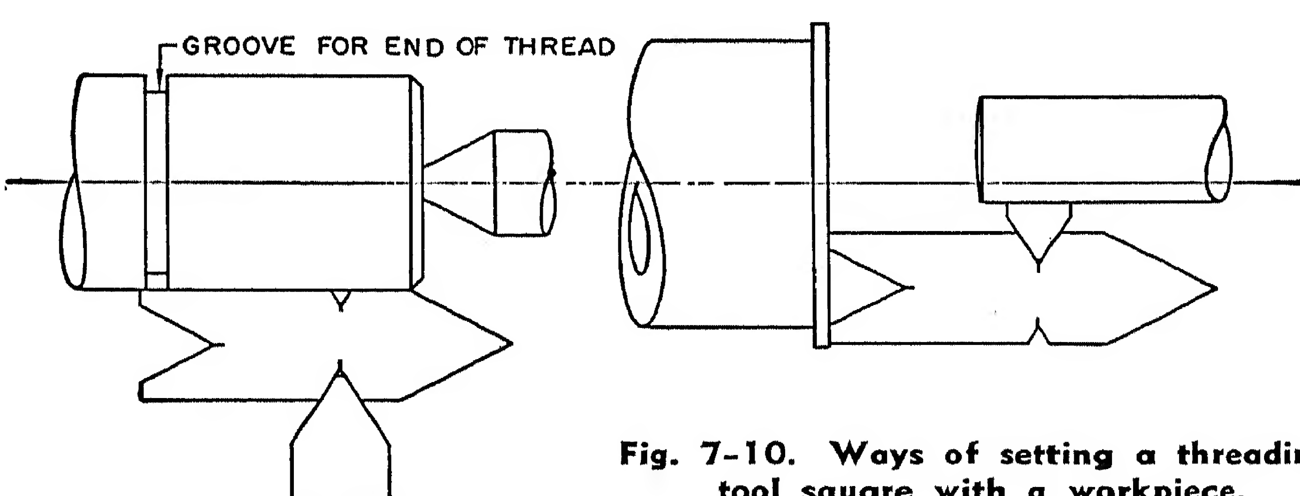


Fig. 7-10. Ways of setting a threading tool square with a workpiece.

7-10. The groove is put in before the thread is started and may be cut with the threading tool or with a grooving tool. The groove has a diameter a little less than the root diameter of the thread, provides a definite guide for stopping the cut, and makes it easier to bring the cutting tool to the end of the thread.

The tool is withdrawn from the thread at the end of each pass, and the carriage is returned to the starting position. The tool must then be reset to the depth to which it has just cut and in addition be set in farther for the next pass. A reference for resetting the tool each time is necessary. That may be obtained at the start by bringing the point of the tool just to touch the workpiece and then setting the cross-feed micrometer dial to zero. Then after the tool has been withdrawn from the cut and the carriage brought back to starting position each time, the cross-feed dial is returned to the same zero position. Some lathes are equipped with a thread cutting stop on the cross slide. It consists of an adjusting screw and block that can be set to position the cross slide in the same place each time. Increments of infeed for successive passes are made with the compound rest feed screw and dial.

Thread chasing is done at ⅓ to ½ of the speed of turning. A light trial cut just deep enough to scribe a line should be taken on the workpiece after the setup has been completed. The number of turns per inch is counted to make sure that the lead is right. Then the tool is positioned for the first cut. The depth of the first cut or two may be as much as 0.005 in., but after that the tool is fed in only 0.002 to 0.003 in. for each cut. A depth of only 0.0005 to 0.001 in. with the tool fed straight in for each of the last few finishing cuts helps produce a clean thread with a good finish. Cutting oil should always be applied freely when chasing threads in steel.

When a single National Form Thread is chased, its crest becomes narrower with each cut and gives an indication of size. Its root diameter may be checked with calipers having narrow points to find out when the finished size is being approached. A Square or Acme Form thread can be measured at the root diameter with calipers. The final check of the fitness of a screw is made with its mating nut or a ring thread gage. An internal thread is checked with a screw or plug gage. The fit should be snug without play or binding in any position.

A left-hand thread is chased by reversing the direction of rotation

of the leadscrew. The workpiece is revolved as always, but the tool travels toward the tailstock.

Tapered screw threads may be chased with the aid of a taper attachment to guide the tool or by setting over the tailstock. In either event, the tool must be set square with the axis of the workpiece.

A multiple thread is cut one space at a time. After the first space has been completed, the work is indexed to the next, and so on. The indexing may be done by transferring the dog on the workpiece from one to another of the equally spaced slots in the dog plate. Another way is to disengage the feed change gears and index the spindle from the teeth of the gears. The spaces of a multiple thread must be all the same size and need to be measured carefully.

After a screw thread has been chased, it is trimmed on the starting end to make it start freely in a nut and to improve its appearance. A 45° chamfer to a diameter a little less than the root diameter of the thread may be put on the end of the piece. Sometimes the whole end of the workpiece is rounded with a forming tool.

Dies and Taps

Dies. A threading die has an internal thread like a nut, but lengthwise grooves in the hole expose the cutting edges of the threads. Dies are made of hardened carbon tool steel or high speed steel. The principal types are called solid, solid adjustable, spring adjustable, and die heads.

A *solid adjustable die* is shown in Fig. 7-11. It is like a *solid die* except that it is split on one side and has an adjusting screw in a cross hole. Whereas a solid die cuts just one size, an adjustable die can be expanded or contracted slightly to change the size to which it cuts. Solid and adjustable dies are commonly sold for standard National Form Thread sizes up to 1½ in. The entering end of the hole in the die is tapered to provide a throat to start the workpiece and distribute the cutting load over several threads. The remaining threads cut to full depth as they pass over the workpiece.

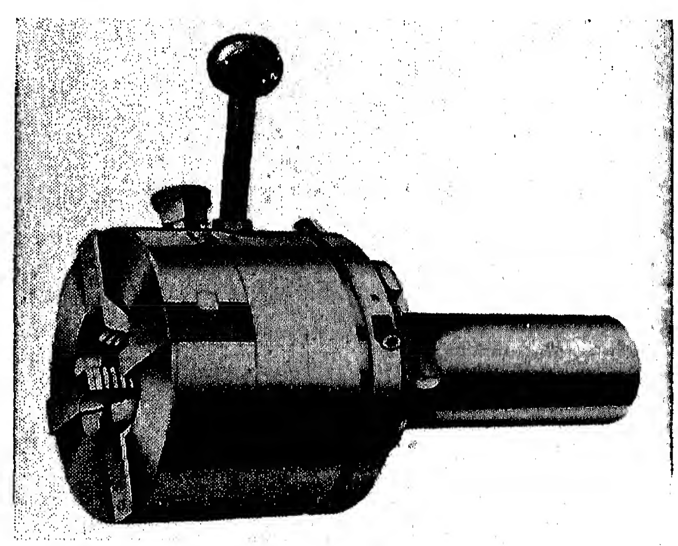
A solid die cannot be sharpened easily and is discarded when it gets dull. Dies of this kind are usually held in a stock, also shown

in Fig. 7-11, and applied by hand. A stock may have a guide or starting sleeve to help start the die square with the work.

Spring adjustable dies and die heads are used on machines, mostly for production. A *spring adjustable screw threading die* is a hollow cylinder with internal threads for part of its length. Its length is greater than its diameter. Lengthwise slits longer than the threaded portion provide openings in the threads to expose cutting edges and allow the die to be sprung like a collet. A collar is clamped around the slotted end of the die to contract the threads and adjust the die to cut to the desired size.

A *die head* has a body in which four or more replaceable and adjustable serrated blades or chasers are mounted. The blades generally are of carbon tool steel or high speed steel. They may be removed when dull, reground, replaced, and adjusted to cut to a desired size. The chasers are mounted radially in some heads. In other heads they are positioned tangentially to the workpiece.

Fig. 7-12. A self opening die head. (Courtesy Geometric Tool Co.)



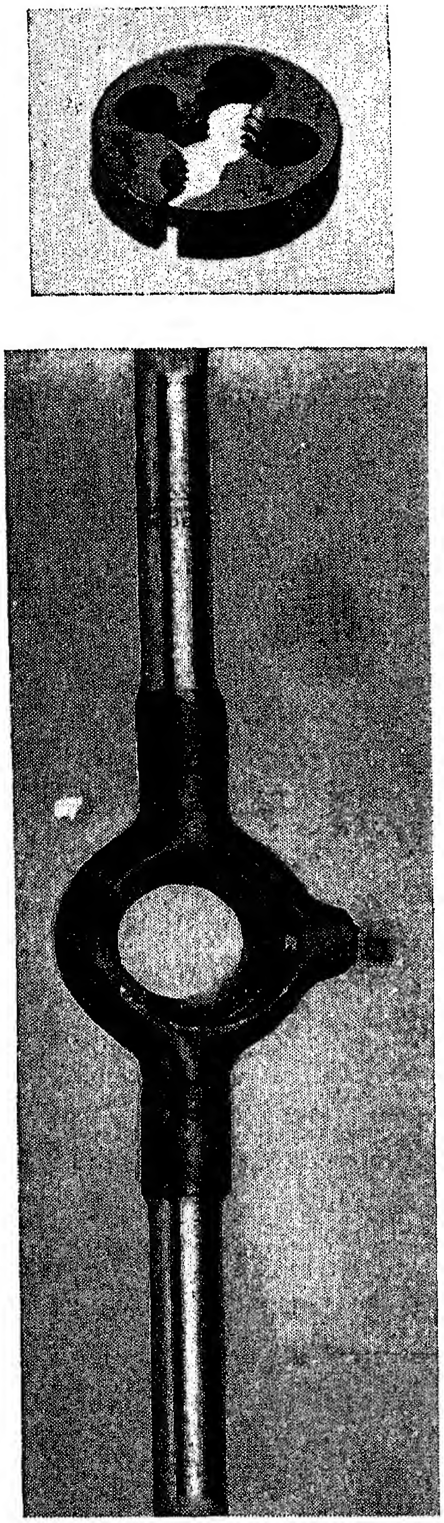


Fig. 7-11. A solid adjustable round split die and stock. (Courtesy Winter Bros. Co.)

A *self-opening die head,* like the one in Fig. 7-12, is arranged so

that its chasers snap outward when a thread has been cut to a predetermined length. In that way the work is released instantaneously and does not have to be screwed out of the die head. A die head on an automatic screw machine or threading machine is usually opened and closed by a preset stop, yoke, or fork attached to the machine. Some heads have an internal trip to contact the end of the workpiece. On a turret lathe the forward movement at the turret is stopped at a predetermined position, but the front part of the die head is pulled along about ⅛ in. more by the chasing action, and that trips the mechanism to open the head. The chasers are restored to cutting position by means of a handle on the die head before another workpiece is cut. Care must be taken to adjust the stops before a thread is chased to prevent the head from being damaged by bumping against a shoulder on the workpiece.

Use of dies on the lathe. A solid or adjustable die is often used to finish a thread that has been chased almost to size on a lathe. The workpiece is held in a chuck. The die is held in a stock and started on the thread by hand. One arm of the stock is allowed to rest against the bed of the lathe. The spindle is turned by hand or very slowly by power. Cutting oil is used, especially for steel.

A die head with a tapered shank may be mounted on the tailstock of a lathe for threading chucked pieces. A die head may also be mounted on a bracket on the lathe carriage. With that arrangement, it is possible to feed by means of the leadscrew and obtain accurate threads.

Threading machines. Universal threading machines like the one in Fig. 7-13 are used for threading bolts, studs, automotive parts, pipes, etc. They are made with one or two heads. Each head carries a revolving self-opening die head with tangential chasers that can be adjusted and replaced for various sizes of threads within the capacity of the machine, for right- and left-hand threads, and for straight or tapered threads. Typical machine sizes are the ½ in. machine for threads from ¼ to ½ in. diameter, the 1 in. machine from ¼ to 1 in. diameter, and the 2½ in. machine from ⅜ to 2½ in. diameter.

Opposite each head on the universal threading machine is a carriage that slides on ways to carry the work to and from the dies A standard vise is normally mounted on each carriage for holding work. The vise may be replaced by a collet chuck or a variety of

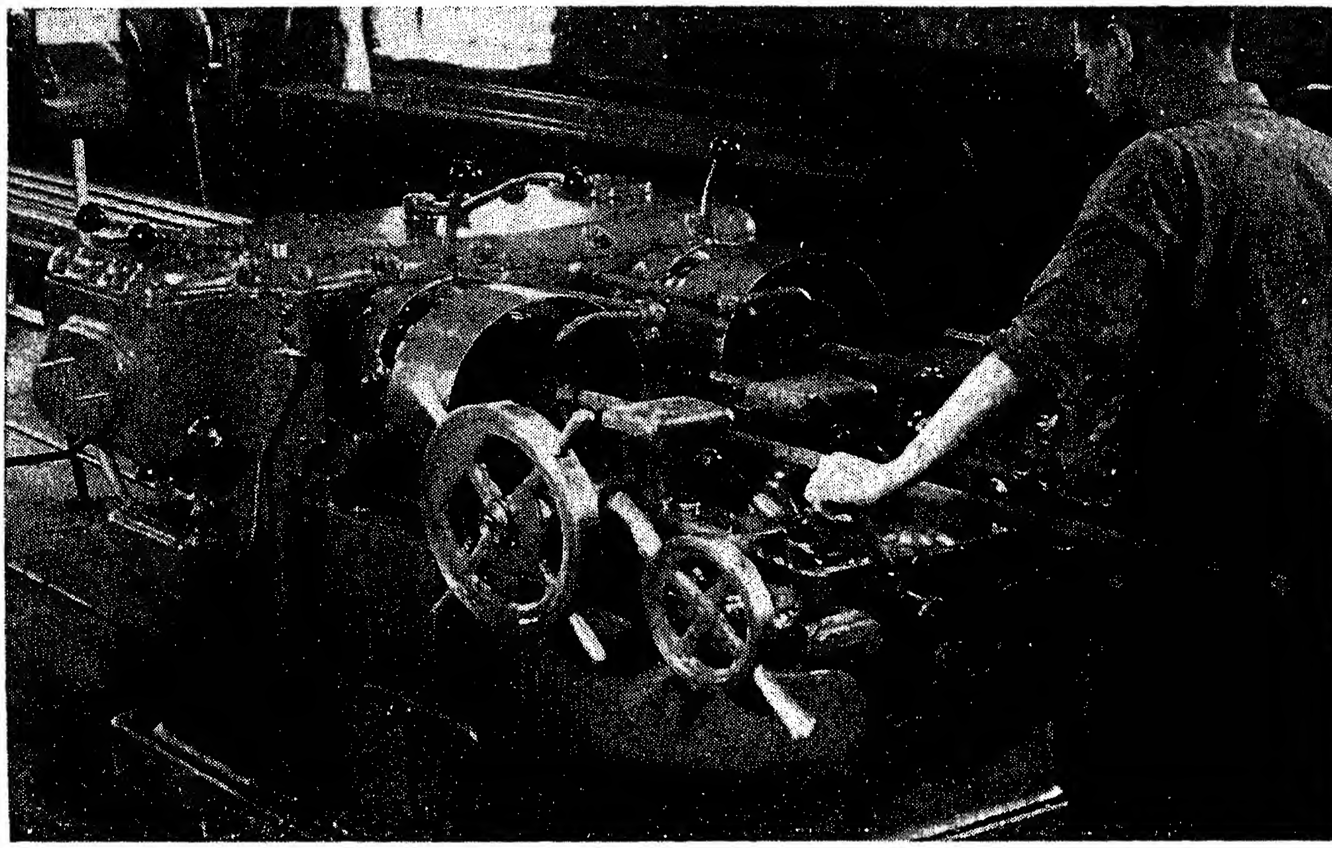


Fig. 7-13. A threading machine with two heads. (Courtesy Landis Machine Co.)

special fixtures. A leadscrew to drive the carriage through change gears from the die head spindle for cutting threads with accurate leads is available as optional equipment. It is engaged by hand but may be disengaged automatically or manually. The two carriages operate independently. The die head is opened automatically at a preset position by the foreward movement of the carriage and is closed as the carriage is withdrawn. The head may also be opened or closed by hand. The machine contains a system for supplying cutting fluid to each head.

Taps. Holes are usually threaded by taps. A *tap* has a shank and a round body with several radially placed chasers. A hole to be tapped is first drilled or bored to leave about 75 per cent of a full thread. Taps are made in many sizes and shapes to satisfy a number of purposes. They may be operated by hand or machine. They are made to cut all forms of threads. Small taps are solid; large taps may be solid or adjustable. A tap has two or more flutes that may be straight, helical or spiral, or spiral pointed. Taps are made of carbon tool steel for low first cost or high speed steel for rapid production and endurance. As a general rule, carbon tool steel taps

and dies on machines are operated at 10 to 20 sfpm, and high speed steel taps and dies at 20 to 60 sfpm. The slower speeds are best for hard materials and small taps, and the faster speeds for soft materials and large taps. Ground high speed steel taps may be run at drilling speeds. A tap works under strenuous conditions because it is buried in metal and fed at an invariable rate. It must be fully

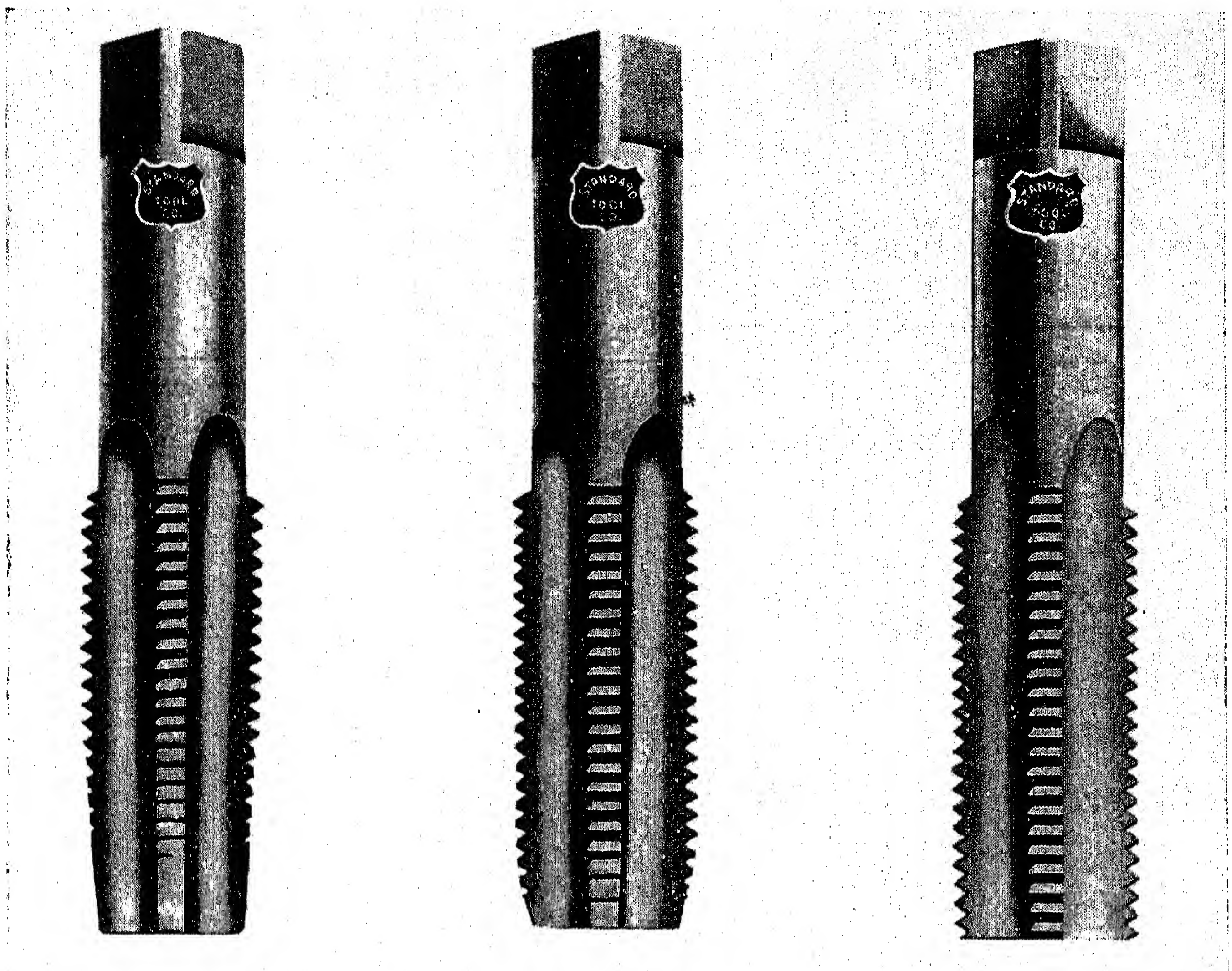
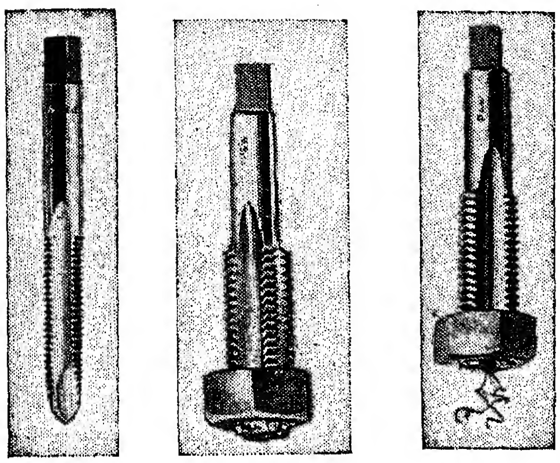


Fig. 7-14. Straight flute hand taps, from left to right; a taper, plug, and bottoming tap. (Courtesy Standard Tool Co.)

supplied with a cutting fluid suitable for the work material to operate successfully. The principal kinds of taps are described in the paragraphs that follow.

Hand taps have short shanks with square ends and are made in sets of three for each size. The three styles are the taper, plug, and bottoming taps, shown in Fig. 7-14. The taper tap enters a cut gradually and is the easiest to start in a hole. A through hole may



A B C

Fig. 7-15. A. A spiral pointed tap. B. The action of a straight flute tap. C. The action of a spiral pointed tap, showing how the chips are driven ahead. (Pratt and Whitney Photo from Pratt and Whitney Division, Niles-Bement-Pond Co., W. Hartford, Conn.)

be tapped entirely by a taper tap, which cuts to full size behind its tapered portion. However, a taper tap will not cut a thread to the bottom of a blind hole. It must be followed by a plug tap, and the latter by a bottoming tap to reach the bottom of a blind hole.

Hand taps are driven by machines as well as by hand. When hand tapping, care must be taken that the tap is started straight with the hole by sighting it square with the surface. The tap need not be forced because the action of the threads draws it into the hole at the proper rate. The tap should be reversed a turn for every two or three forward turns to clear the chips and prevent binding. This is particularly important for small taps because they can be broken easily if stuck in the hole. Cutting fluid should always be used freely when tapping—cutting oil for ferrous materials, kerosene for aluminum.

Alignment with the hole is also important in machine tapping. A floating tap holder should be used if alignment is not assured. Frictional tap holders are used on machines because they can slip if the going becomes hard or the tap strikes the bottom of the hole. The holders avoid tap breakage.

Small taps usually have two or three flutes, and those above ½ in. diameter have four flutes. A small number of flutes allows a large flute space. Most taps have straight flutes, but some have helical flutes. A flute at a right angle to the helix of the thread exposes the true thread form. Steeper helix angles help to clear away chips. *Spiral pointed taps,* as shown in Fig. 7-15, have a negative helix on the end along the tapered teeth. This serves to drive the chips ahead of the tap and to prevent their clogging the flutes.

Serial hand taps are made in sets of three. Two are undersize for roughing and are used first. A serial tap has one or more rings scribed around its shank, and the number of rings designates its

place in the use sequence. Serial taps are used in tough metal and produce smooth threads.

A *pulley tap* is like a hand plug tap but has a long shank of the same diameter as the minor diameter of the thread to act as a guide. Pulley taps are used to reach holes that are somewhat inaccessible, like a set screw hole in the hub of a pulley.

A *nut tap* has a long shank with a square tang. A small point on the fluted end enters holes readily, and a long chamfer serves

A
B
C

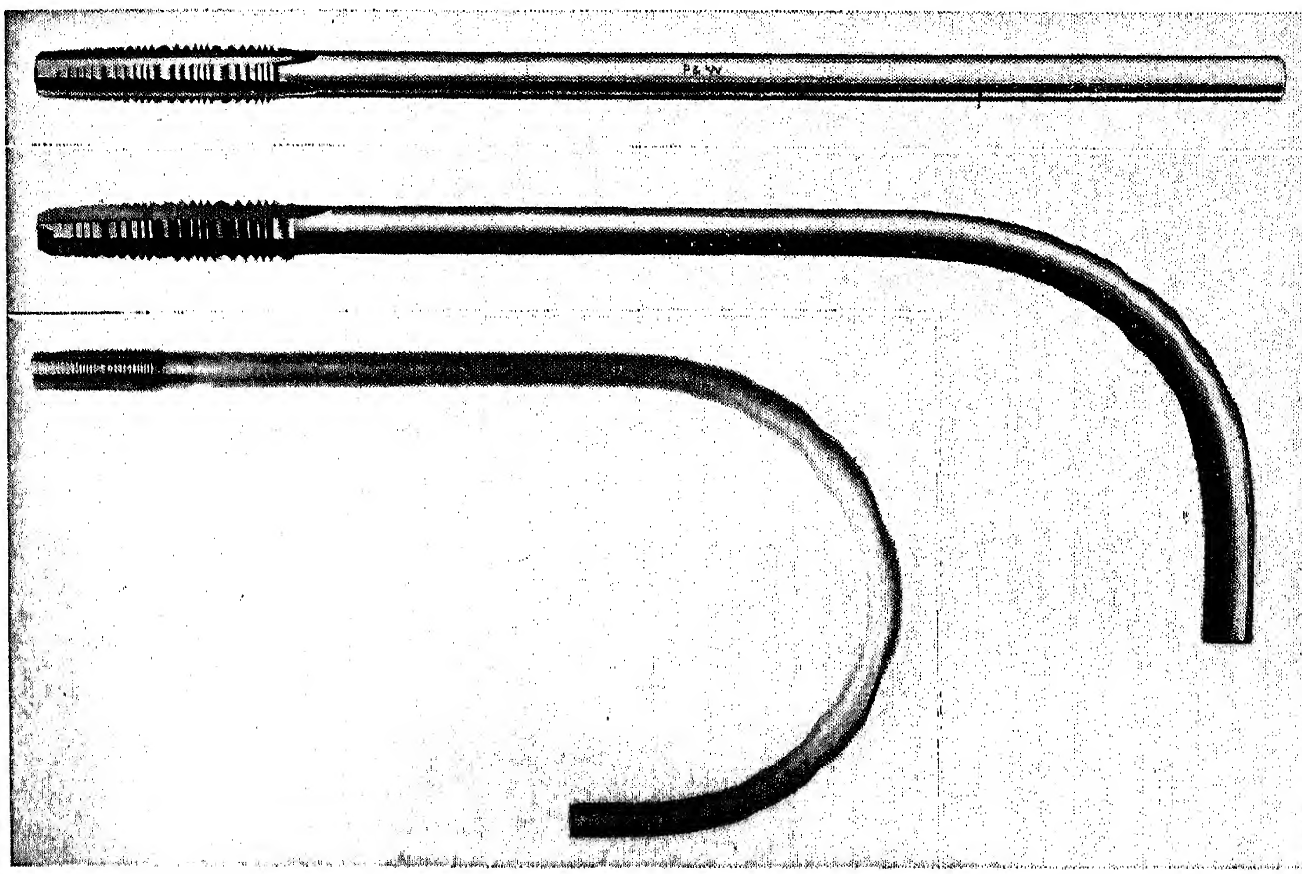


Fig. 7-16. Tapper taps. (top) **Straight shank tapper tap. (Pratt and Whitney Photo from Pratt and Whitney Division, Niles-Bement-Pond Co., W. Hartford, Conn.)** (middle) **Bent shank tapper tap.** (bottom) **Hook shank tapper tap. (Courtesy The National Machinery Co.)**

to distribute the cutting action over many threads. Nut taps are used in nut tapping machines and drill presses for through holes. They are favored for tough materials, small quantity production, and where a long shank is required.

Tapper taps are used for tapping nuts in large quantities on specialized nut tapping machines. They are made in standard sizes up to 2 in. diameter. *Straight shank tapper taps,* as illustrated in Fig. 7-16 A, have length from 6 to 15 in. and various kinds of shanks such as plain round, flattened, and square. The thread length

is not tapered like that of a nut tap but is simply chamfered at the point. The shank is smaller than the minor diameter of the thread. The *bent shank* and *hook shank tapper taps* of Fig. 7-16 are designed for high production tapping on automatic machines like the one depicted in Fig. 7-18.

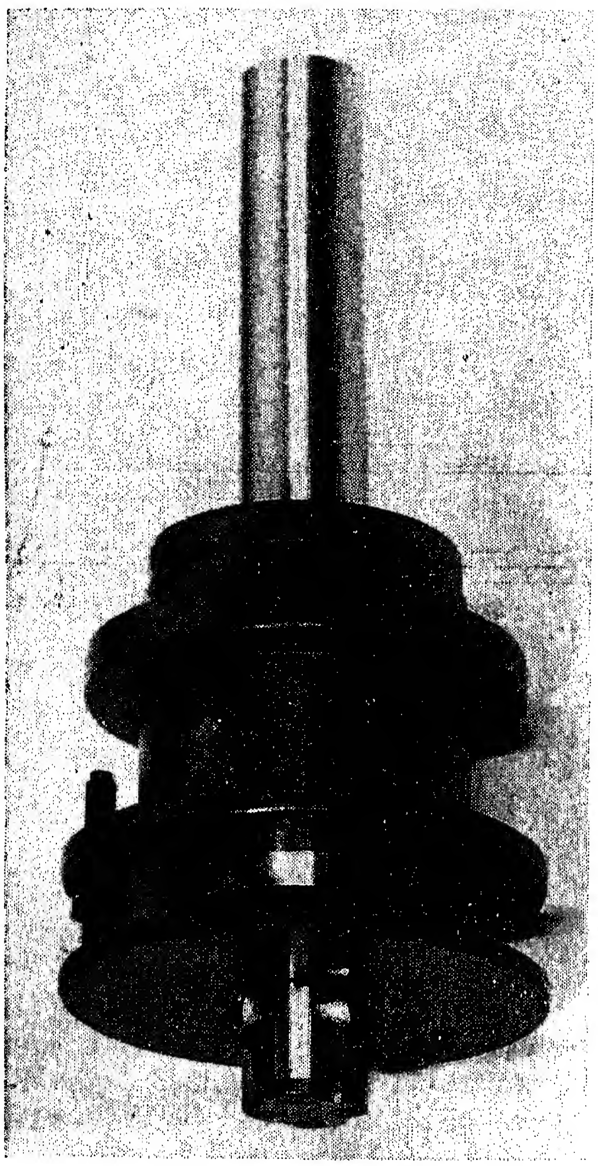


Fig. 7-17. A collapsible tap. (Courtesy Murchey Div., The Sheffield Corp.)

Collapsible taps operate like self-opening die heads and are used on turret lathes and automatic screw machines. Threads are cut by several chasers held in the body. The collapsible tap of Fig. 7-17 has two collars behind the chasers. The front collar may be adjusted to trip the tap from the face of the work when threads have been cut to depth. A finger yoke on the machine may also serve to trip the tap by pressing against the rear collar. When the tap is withdrawn, the finger presses against the rear face of the front collar and resets the tap. A handle like the one on the die head of Fig. 7-12 is provided on some collapsible taps for resetting them.

Pipe taps are tapered and are used to cut internal pipe threads that are tapered. One style carries a short drill in front of the tap to clean out the hole to be tapped.

Acme taps cut internal Acme threads and must remove a large amount of metal. If one tap is used, its tapered and threaded portion is normally quite long. Instead of a single long tap, Acme taps often are made in sets consisting of several roughing taps and a finishing tap.

Tapping on the lathe. A workpiece normally is chucked when tapped on a lathe. The shank end of the tap is supported on the tailstock center. A wrench on the square tang of the tap is kept from turning by the carriage. The work spindle is revolved slowly

by power or hand, and cutting oil is flowed on the tap.

Tapping machines. Much tapping is done on drilling machines. Even hand tapping is commonly done on the drill press. The workpiece is clamped in place, and the hole is drilled. The drill is withdrawn, taken from the spindle, and replaced by a center, but the workpiece is not moved. A tap is started in the hole and is guided by the center in the drill press spindle. The center is placed in the center hole in the shank end of the tap and aligns the tap true with the hole.

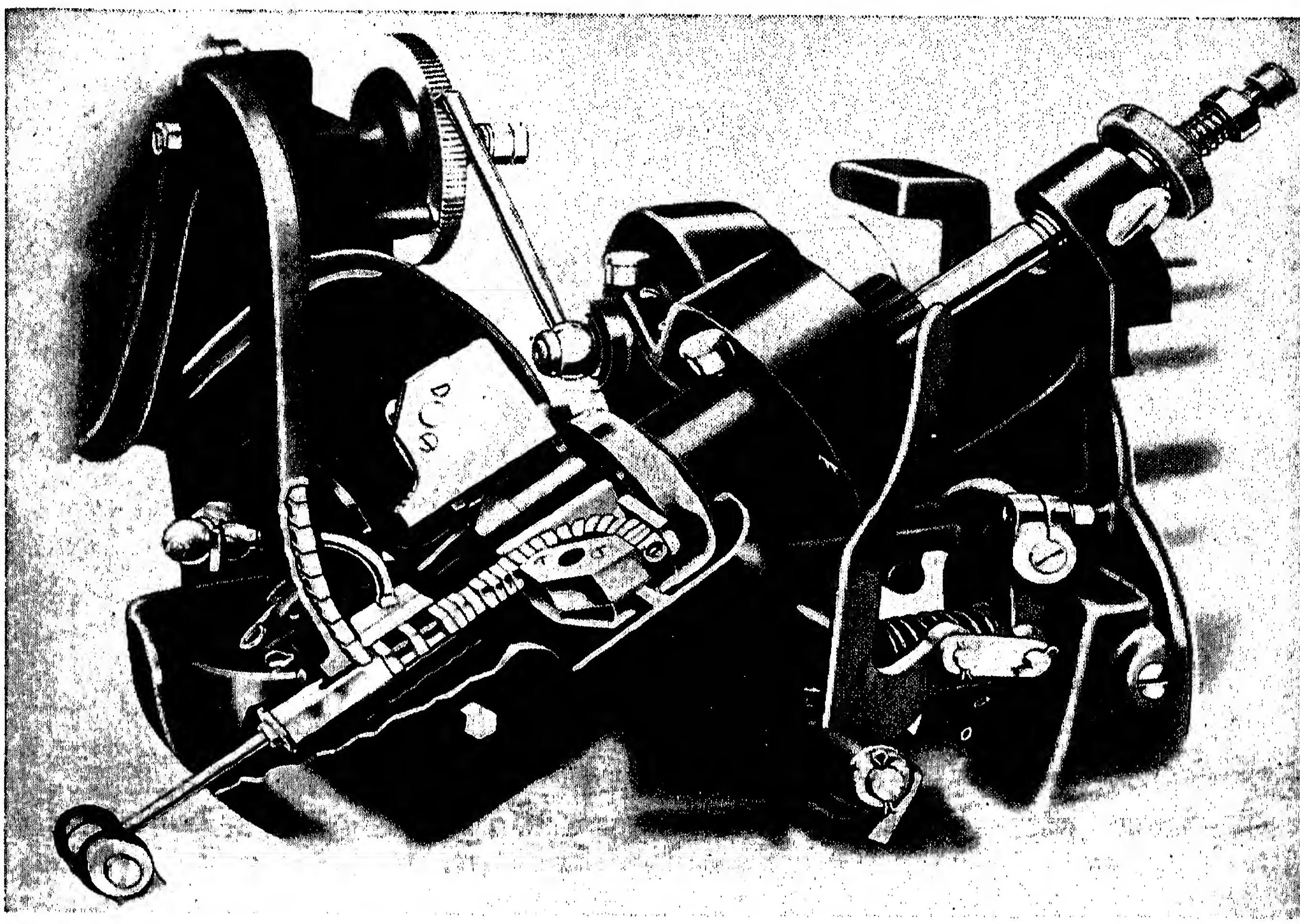


Fig. 7-18. A sectional view of the head of an automatic tapper that uses a bent tap. (Courtesy The National Machinery Co.)

Some tapping machines are basically drill presses equipped with tap holders, reversing mechanisms, leadscrews, etc. to enhance their tapping ability. They may have one or more spindles. Drilling machines are described in Chapter 11.

A tapping attachment may be fastened to the spindle of a standard drilling machine that does not have built-in tapping accessories. Some attachments are of the speed-up type and revolve small taps

at high speeds. Tapping attachments are arranged to rotate the tap in a forward direction when it is lowered into the hole and reverse the direction of rotation when the tap is withdrawn from the hole. This action may be obtained from opposed sensitive friction clutches in the tapping attachment head.

Machines for tapping nuts and other small parts are made with four, six, or eight vertical spindles in a row. The nuts are placed in fixtures under the spindles. The taps are lowered at a rate equal

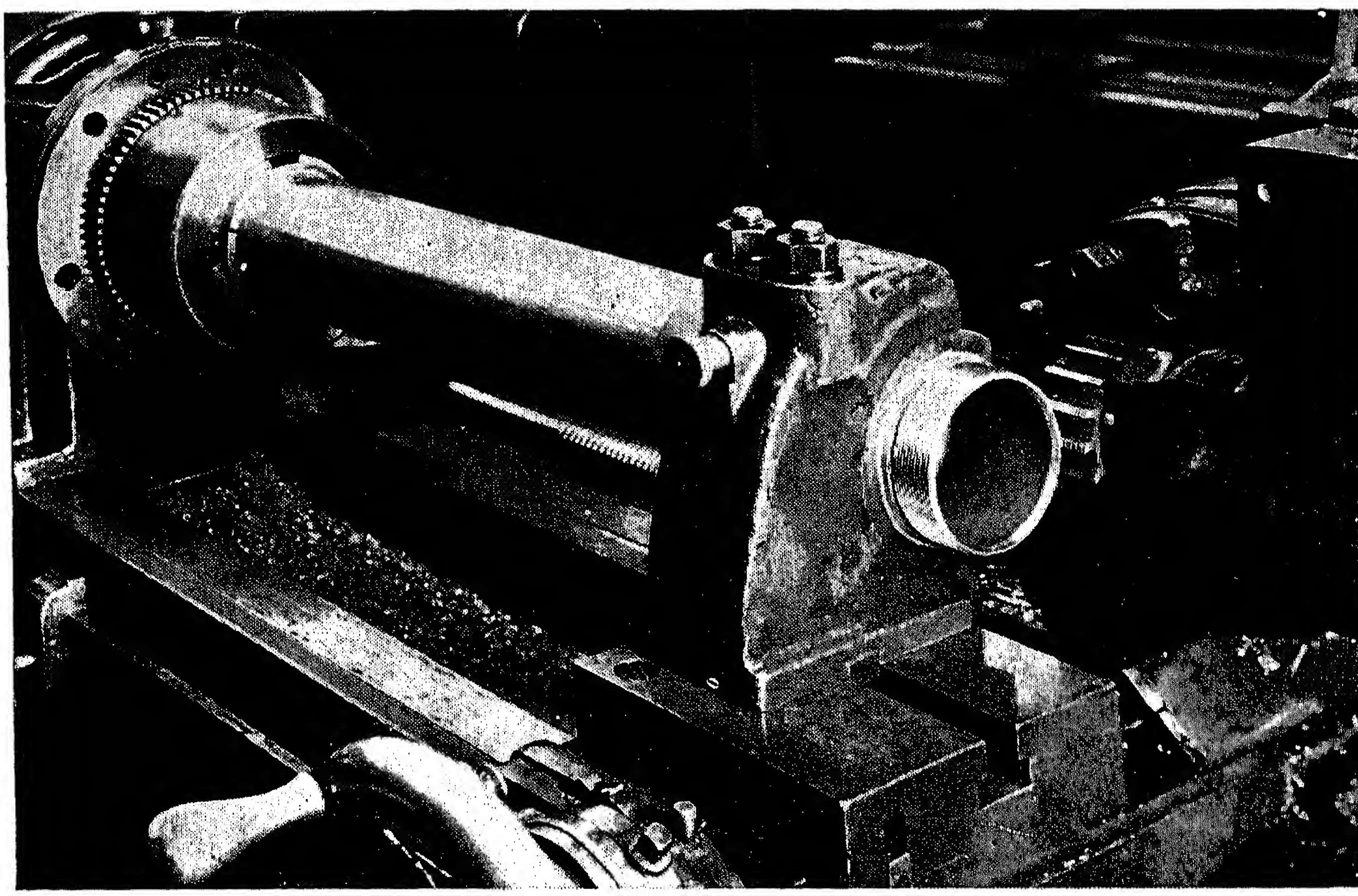


Fig. 7-19. A view of an external thread milled with a multiple thread form cutter on a thread miller. (Pratt and Whitney Photo from Pratt and Whitney Division, Niles-Bement-Pond Co., W. Hartford, Conn.)

to the lead of the thread. The spindles advance one after another on one style of machine, and each station is loaded while its spindle is in a raised position. Straight shank tapper taps are used. Nuts collect on each tap shank, the tap is removed from the spindle when full, and the nuts are stripped off. Another style of machine has two sets of fixtures, and one set is unloaded and loaded while the workpieces in the other set are being tapped. The taps are advanced all at the same time, reversed at the end of the cuts, and screwed out of the workpieces and retracted at a fast rate. Pro-

duction rates of as high as 150 pieces per minute are achieved on multiple spindle tapping machines.

An illustration is given in Fig. 7-18 of the mechanism of an automatic nut tapper that uses a bent tap to produce nuts continuously. Blanks pass from a hopper down a chute and enter an ejector that feeds them onto the tap one at a time. The tap is revolved continuously and is moved forward into each blank. The nuts are held in guides to keep them from turning and are not pulled along by the tap while the threads are being cut, to prevent binding. After one nut has been tapped, the tap with the nut on it is moved back, ready for the next blank. The tap is always filled with nuts, which support and hold the tap central in the revolving head. As each nut climbs onto the tap, it forces another off the bent end.

A conventional thread milling machine is like a lathe with a headstock and tailstock mounted on a bed. A carriage slides between the two on ways and carries a cutter head. The work may be mounted between centers or chucked on the headstock spindle. The workpiece in Fig. 7-19 is chucked and also supported by a steady rest on the carriage.

Multiple thread form cutters, like that shown in Fig. 7-19, are used for rapid thread cutting. The cutter does not have a thread or lead. Instead its teeth are arranged on a series of closed circles. The axis of the cutter and work are in parallel planes but at an angle equal to the helix angle of the thread. The revolving cutter on the head on the carriage is fed into the work to depth. The cutter is longer than the thread. As the work revolves, the cutter is advanced lengthwise an amount equal to the pitch of the thread during one revolution of the workpiece. At the end of 1 1/10 revolutions of the work, the cutter is withdrawn from the completed thread. An internal thread may be milled in a similar manner. Thread milling as just described is often as fast as thread cutting with self-opening dies and collapsible taps and produces more accurate threads and better finishes. It is confined mostly to V-type threads because too much form error is introduced in other threads.

Coarse threads, like those on feedscrews and leadscrews, are milled with single cutters on a *universal thread miller.* Its cutter head can be tilted to incline the cutter axis to match the helix angle of the thread. As the work revolves, the cutter head and carriage are moved longitudinally on the machine by a leadscrew or cam

to produce the desired lead. The cutter thus traverses the entire thread, and the length of thread is limited only by the capacity of the machine.

Threads can also be milled on a planetary milling machine described in Chapter 12.

Thread Rolling

Thread rolling is a cold forging rather than a cutting process. It produces external threads by subjecting a blank to pressure between steel dies or rolls that are ridged or threaded. The initial diameter of the blank is approximately the same as the pitch diameter of the thread formed. The work material is depressed to open the root and raised to form the crest of the thread.

Two flat-faced dies like those in Fig. 7-20 are mounted in a reciprocating-type machine to roll threads by the *flat die method.* One of the dies is held in a fixed position, and a blank is started at one end of its face. The other die is forced lengthwise across the first die with the blank between the two faces. The workpiece rolls as it is squeezed between the two dies, and threads are impressed on it by the serrations in the dies. When the moving die has passed the fixed die, the completely threaded piece is released.

Some thread rolling machines employ grooved rollers. Two rollers are used in one type of machine on which the workpiece is supported on a workrest blade between the rollers. In another type of machine, three rollers surround the blank. The rollers are forced in toward the center of the blank as they rotate at uniform speed. At the end of the cycle, the rollers move apart to release the threaded piece.

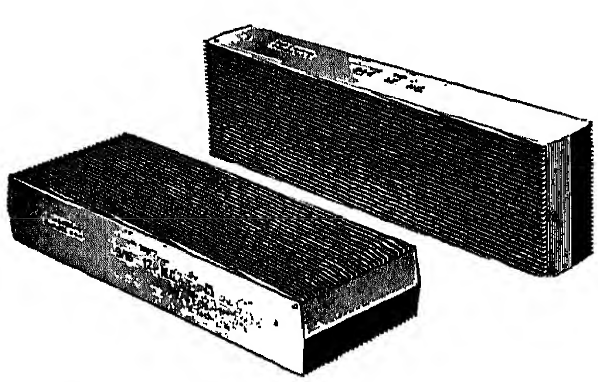


Fig. 7-20. A pair of thread rolling dies. (Pratt and Whitney Photo from Pratt and Whitney Division of Niles-Bement-Pond Co., W. Hartford, Conn.)

Thread rolling is capable of producing accurate, uniform, and smooth threads at high rates of production on all kinds of screws, bolts, and studs. It can be adapted to making almost all screw thread forms. It is economical of material because none is cut away to form the spaces. The material is cold worked, and its physical properties are improved. Output

from 30 pieces per minute on large parts to 175 pieces per minute on small parts is attained.

Thread rolling is not suitable for threading pieces in small quantities and is essentially a high production process. Initial blank diameters must be controlled, and the thread diameter must be machined to a smaller size than the shank if the finished thread is to have the same diameter as the shank of the screw. Under certain conditions, bending of the blank, seams, and slivers on the thread result from rolling. The last fraction of a turn on the end of a rolled thread is truncated.

Questions

1. Define a screw thread, a right-hand thread, and a left-hand thread.
2. State three general uses of screw threads and give an example of each.
3. What is meant by the pitch and lead of a screw thread? How do the pitch and lead agree for a single thread screw and for a multiple thread screw?
4. Define the major, minor, and pitch diameters of a thread.
5. Make a sketch of a Unified Screw Thread Form and name its parts.
6. Make a sketch of an American National Acme Thread Form and name its parts.
7. What does the Unified Screw Thread Form Standard specify?
8. Describe four ways of measuring or checking screw threads.
9. What are the advantages and disadvantages of chasing a thread on a lathe?
10. Describe the principal ways of making screw threads.
11. How is an engine lathe set up and operated for chasing a thread?
12. Describe the principal types of dies. How are they used?
13. Describe the three styles of hand taps. How are they used?
14. What are serial hand taps, pulley taps, nut taps, tapper taps, collapsible taps, and pipe taps?
15. Describe three kinds of tapping machines.
16. In what two ways may threads be cut on a thread milling machine?
17. Describe the process of thread rolling. What are its advantages and disadvantages?

Problem

1. The leadscrew on a lathe has four threads per inch. Specify the gear ratios required to cut each of the following numbers of threads per inch: (a) 4 (b) 5 (c) 6 (d) 7 (e) 8 (f) 12 (g) 16 (h) 18 (j) 20 (k) 25.

References

Boston, Orlan W., *Metal Processing.* New York: John Wiley and Sons, Inc., 1951.

Gage Blanks, Commercial Standard CS8–41. Washington, D.C.: U.S. Department of Commerce, 1941.

How to Run a Lathe. South Bend, Ind.: South Bend Lathe Works.

Pipe Threads, American Standard ASA B2.1–1945. New York: American Standards Association.

Screw Threads and Tap-Drill Sizes, Commercial Standard CS24–43, Washington, D.C.: U.S. Department of Commerce, 1943.

Screw Threads Standards for Federal Services, Handbook H–28, Washington, D.C.: U.S. Department of Commerce, National Bureau of Standards, 1944.

Tool Engineers Handbook, A.S.T.E. New York: McGraw-Hill Book Co., 1949.

Unified and American Screw Threads. Proposed American Standard ASA B1.1–1949. New York: American Standards Association.

Manufacturers' Catalogs:

Bath, John, and Co., Inc., Worcester, Mass.
Brown and Sharpe Mfg. Co., Providence, R. I.
Geometric Tool Co., New Haven, Conn.
Landis Machine Co., Waynesboro, Pa.
Murchey Machine and Tool Co., Dayton, Ohio.
National Machinery Co., Tiffin, Ohio.
Pratt and Whitney Division, Niles-Bement-Pond Corp., Hartford, Conn.
Standard Tool Co., Cleveland, Ohio.
Winter Bros. Co., Wrentham, Mass.

Chapter 8

PROCESS PLANNING

EVERY OPERATION THAT IS DONE ON MACHINE TOOLS PRESENTS A NUMBER OF PROBLEMS. They must be solved to find the best way to meet the following three basic requirements:

1. A piece or pieces of definite form and shape must be produced.
2. Specified limits of accuracy for linear and geometric dimensions and for quality of surface finish must be held.
3. The rate of production must be as high as possible with the facilities available or justified; the job must be done at the lowest possible cost; and the product must be ready when needed.

Operation Planning

Operation planning as an engineering function. The engineer's role is to plan manufacturing processes and operations to satisfy these requirements. That starts with the design of a product. A part can usually be designed in several ways to suit a purpose. The designer must be able to visualize the steps and cost to make each possibility so that he can select the most economical. After a part has been designed, preparations must be made to produce it. In planning for production, an engineer selects tools and equipment, decides upon the arrangement of the equipment and operations, and estimates the cost. The purpose of this study is to introduce the principles that enable the engineer to solve these eproblems systematically and effectively.

What a metal machining operation is. The steps leading to the completion of a manufactured article constitute a *process*. Certain steps in a process are called *operations*. Generally an operation in

metal machining is expected to include at least enough activity to produce a distinct surface on a workpiece. Thus, the drilling of a hole may be considered an operation, even though the hole is later to be bored or reamed. On the other hand, merely starting the drill or setting the speed of the machine would not be considered an operation. Such steps together with others make up the operation of drilling the hole and may be called *elements* of the operation.

The smallest units of accomplishment worthy of being called operations may be termed *basic operations.* They are grouped into *major operations* when done together. As an example, a major operation may consist of facing the end of and center drilling, drilling, boring, and reaming a hole in a workpiece. Both basic and major operations are commonly referred to as operations.

A major operation customarily includes all the work done on a piece in one setup on a machine. Consider a workpiece chucked on a lathe to have a hole finished. The basic operations of, say, core drilling and reaming make up one major operation. Then the workpiece is mounted on an arbor, and its outside surfaces are turned between centers. That calls for a new setup of the lathe and a second major operation. Finally, the outside diameter is finished on a grinding machine. Again a new setup is required, this time on an entirely different machine, and a third major operation is constituted.

Where a number of pieces of one kind is processed, all in one lot normally are put through each major operation while the machine is set up for it. Suppose five pieces are to be machined on the surfaces described in the preceding paragraph. The operations could be arranged in a number of ways, but only one way is the most efficient under the circumstances. For instance, each piece could be chucked, core drilled, and taken out of the chuck. Then each piece could be rechucked in turn and reamed. That means two major operations instead of the one originally proposed. The single major operation for drilling and reaming is better because less time is lost in handling the workpiece. Also, errors are introduced each time a piece is rechucked. For accuracy, as many cuts as possible should be taken on a workpiece in one setting.

Another possible arrangement of the process under consideration is to combine all the lathe work into one major operation. For five pieces, that would involve mounting the chuck on the machine, chucking the piece, finishing the hole, taking off the chuck, setting

up for straight turning the outside, and tearing down the setup for each piece. That would mean tearing down and setting up five times for the one lot, which is obviously wasteful.

The same process for five pieces might be looked upon as one major operation and be arranged so that the operator would carry each piece from machine to machine, completing it before starting another. That would require extra time to move from machine to machine. Also the operator would have to exercise versatility and would be prevented from concentrating on one task.

These considerations lead to the principle that for a number of pieces of one kind, a major operation normally includes all the work logically done on one piece in one setup on one machine.

The steps in operation planning. Operation planning is not difficult even for complicated parts if it is done methodically. Many factors have to be considered to arrive at an efficient plan for each case. They may seem overwhelming if viewed as a whole, but they can be disposed of easily if taken one at a time. The problems in planning for metal machining operations are most readily solved step by step, detail by detail, in the same way that an intricate mechanism is assembled piece by piece.

The procedure for planning a metal machining operation is the following:

1. Determine what must be done from the specifications and requirements of the job.
2. Determine how the workpiece must be treated to fulfill the specifications and requirements.
3. Select the tools, machine, and other equipment that most efficiently provide the prescribed treatment.
4. List the steps to be taken in performing the operation and arrange them in sequence to utilize the tools, machines, and equipment most efficiently.
5. Estimate the time required to perform the operation as planned.

This procedure will be explained and illustrated. The examples will be confined as much as possible to the tools and machines so far presented. However, the principles apply to all kinds of metal machining and should be applied by the student in planning operations on other machine tools when he becomes familiar with them.

To Determine What Must Be Done

Requirements and specifications. The first consideration in solving any problem is to find out what its conditions are and what is wanted. The three basic requirements that have been given are conditions that apply to all metal machining operations. Requirements that vary and must be ascertained for each operation are:

1. The size, shape, and other features of the workpiece as it comes to the operation
2. The specifications of the workpiece that must be produced by the operation
3. The number of pieces to be made

Workpiece specifications often are given by drawings, like Fig. 8-1. They are called *part drawings* when they depict finished pieces. *Operation drawings* often are made to show specifications for individual major operations.

The information that makes up the specifications of an operation includes:

1. The sizes and geometric forms of the surfaces to be machined. This includes the dimensions between surfaces.

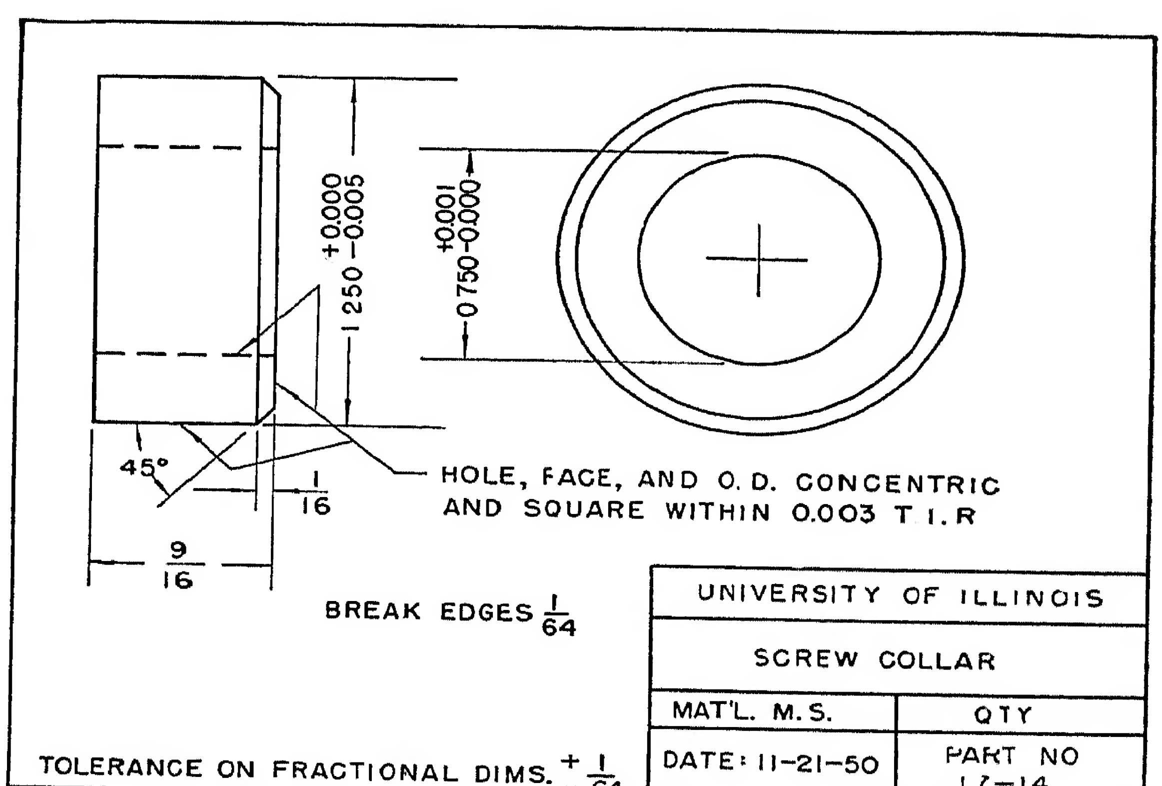


Fig. 8-1. A typical part drawing.

2. The tolerances of linear and geometric dimensions and surface finish
3. The amount of stock to be removed by machining
4. The kind of workpiece material and its physical properties

The part illustrated in Fig. 8-1 will be used as an example for all the steps of operation planning. For this purpose, one piece is needed and is to be made from a piece of bar stock 1½ in. diameter by 3 in. long. The size and shape of the workpiece is shown in the drawing of Fig. 8-1, which specifies the following:

1. An outside surface with $1.250 \begin{smallmatrix} +0.000 \\ -0.005 \end{smallmatrix}$ in. diameter
2. A hole with a $0.750 \begin{smallmatrix} +0.001 \\ -0.000 \end{smallmatrix}$ in. diameter
3. Two plane parallel surfaces 9/16 in. apart
4. A $45° \times 1/16$ in. chamfer on one outside edge
5. The hole, one face, and the OD to be concentric and square within 0.003 in. total indicator reading
6. The material to be machine steel

Some of the specifications are not given directly on the drawing. That is often the case. Such information must be deduced from other facts or be obtained by inquiry. For instance, the amount of stock to be removed from the outside diameter is found by subtracting the finished size of 1.250 in. from the stock size of 1.500 in. and is 0.250 inch. The amount of stock to be removed from the faces needs to be only enough to clean up the surfaces and give the tools sufficient metal to cut. That is usually about 1/16 in. on each outside face. The stock allowed for cut-off must be sufficient to permit use of an adequately strong cut-off tool.

The quality of surface finish expected is not specifically stated on this drawing and often is not specified on drawings. That is because in most cases the normal finishes produced by machining are acceptable. If that is the case, it should be clearly understood.

The Treatment of the Workpiece

In the preceding step, the requirements and specifications were noted but nothing was said about how they would be realized. The specifications and requirements tell what must be done to a work-

piece but not how it should be done. In the second step, the feasible ways of treating the workpiece surfaces to realize the required results are considered. This means visualizing how the tools should act upon the surfaces to be machined and what surfaces should be used for holding and locating the part. This consideration does not call for the selection of specific tools and machines to do the job. Rather the basic machining methods are chosen now, and the equipment needed to carry them out effectively is the next consideration. In the event the best method is not readily discernible at this stage, it may be necessary to select several promising methods, carry them through succeeding stages, and evaluate each in its practical aspects to find the most desirable.

Appraisal of the basic machining methods. Surfaces are machined in a relatively few basic ways. Some of them are depicted in Figs. 1-1, 1-2, and 1-3. Others will be seen as more machines are studied. A number of machine tools is available to carry out each basic method. For instance, turning involves rotating a workpiece and traversing a tool in a definite path. That can be done by several kinds of lathes and turning machines, each of which does the job best under certain circumstances. The logical approach is to narrow the choice to a desirable basic method and then find the equipment to carry out that method efficiently.

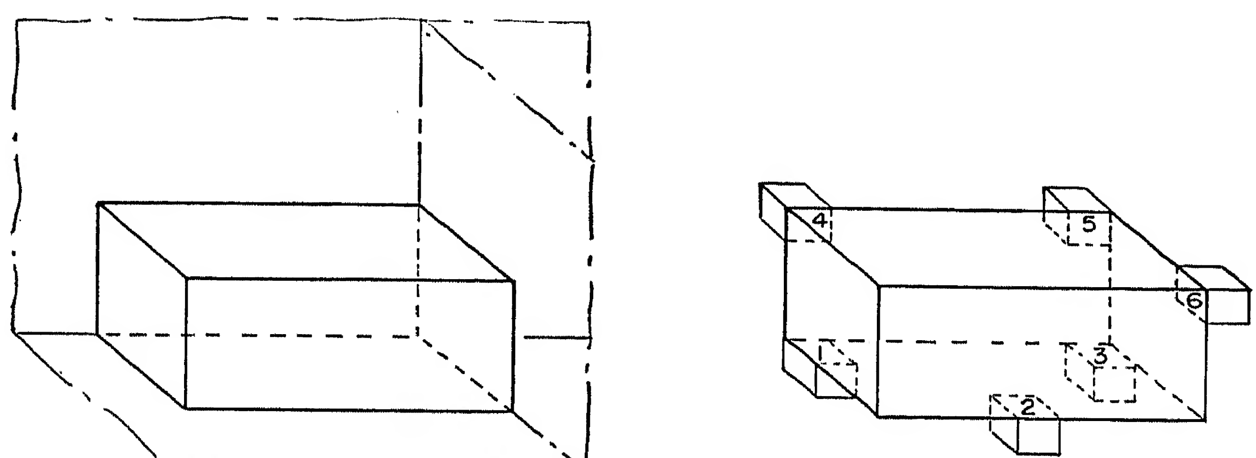
The outside diameter of the screw collar of Fig. 8-1 can be turned in the manner indicated in Fig. 1-1 A. It can also be machined by revolving the tool instead of the workpiece or by feeding a wide tool radially into the revolving workpiece. The latter two methods require somewhat unusual tools and are not favored for producing pieces in small quantities. Two turning cuts are needed to remove ¼ in. of stock on the diameter and produce acceptable finish and accuracy.

The hole in the part of Fig. 8-1 must be drilled from the solid, bored to make it concentric with the outside, and reamed to the tolerance required. Those operations can be done by revolving either the workpiece or tools. The workpiece will be revolved for turning the outside surface, so expediency dictates that it also be revolved for working the hole. The faces could be faced, shaped, or milled as indicated in Figs. 1-2 and 1-3, but facing is the logical choice to go along with the other operations. The finished piece must be cut from the bar from which it is made. It can be sawed

off or cut off by a single point tool after the other operations have been done.

Thus the conclusion is reached that the proper treatment of the part of Fig. 8-1 is to revolve the bar and feed the tools across the surfaces to be machined.

Locating and holding the workpiece. When a workpiece is fully located, it occupies a definite position on the machine with respect to the tools and can be treated in a definite manner. What is more important is that any other workpiece of the same kind can be likewise located in the same way and receive the same treatment. That



A. LOCATION FROM 3 PLANES

B. LOCATION FROM POINTS IN 3 PLANES

Fig. 8-2. A fully located block.

is one of the principles that makes possible the manufacture of duplicate parts.

A piece must be held against three planes or points in such planes to be fully located. An example of a block fully located in that manner is shown in Fig. 8-2. Planes that are square with each other are best but are not altogether necessary, especially if the workpiece surfaces are oblique with each other. Also two or three parallel planes may serve as well as any one of the primary planes.

The number of points required to locate a part fully is prescribed by the 3-2-1 principle of location. It states that *a part can be located fully by placing and holding it against three points in a base plane, two points in another plane, and one point in a third plane,* when the planes are not parallel and are preferably square with each other. More points than prescribed do not add to the effective-

ness of location and may impair it. The application of this principle to a block is shown in Fig. 8-2. The points 1, 2, and 3 fix the base plane, points 4 and 5 the second plane, and point 6 lies in the third plane. Figure 8-3 gives an example of the use of the principle for an object of another shape. In that illustration, the points 1 and 2 in one plane and 4 and 5 in another plane locate the cylindrical surface of the part as fully as needed.

Full location is not always necessary or provided. All surfaces of the part of Fig. 8-3 are definitely located because point 3 fixes the radial position of the head, and point 6 determines the axial position of the part. A cylinder is located from four points in Fig. 8-4 and has two

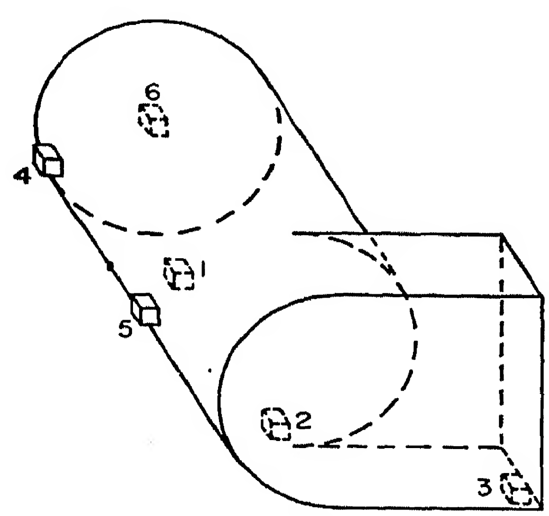


Fig. 8-3. Full location of an irregular object.

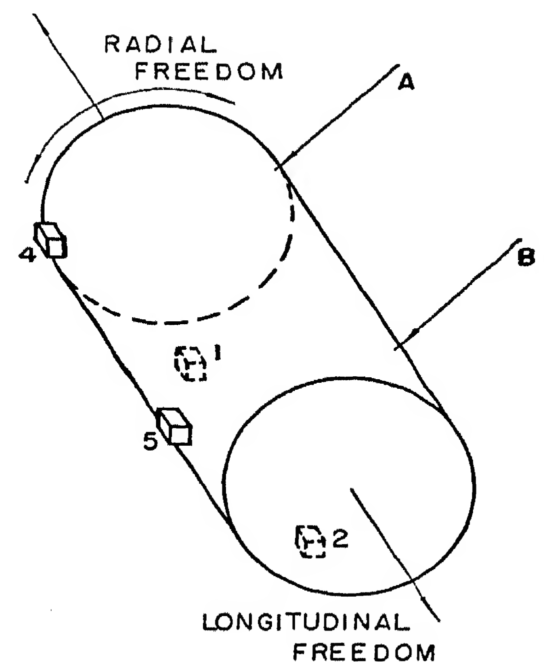


Fig. 8-4. A round piece located by four points.

degrees of freedom. The points 1, 2, 4, and 5 are sufficient to locate the axis of any cylinder of the same size. No means are provided to locate the cylinder around its axis or endwise. This is similar to locating a round piece in chuck jaws or in a V-block fixture. A piece between centers has one degree of freedom because it is still free to turn. In many such cases, the chief concern is to be able to rotate the workpiece around a definite axis, and that is accomplished by partial location.

As a rule, a number of surfaces and points on a workpiece are available to satisfy the 3-2-1 principle of location. Several considerations are helpful in selecting the most desirable spots. These are:

1. The full area of a finished surface may serve very well for

location, but only the least required number of points on a rough and uneven surface are desirable.

2. Locating points on a surface should be placed as far apart as possible to minimize errors.
3. The best surfaces for location are those that are connected by dimensions to the surfaces to be machined in an operation.
4. Locating points should be placed where they serve best to support the part against cutting forces, where other considerations are not impaired.
5. Surfaces to be machined obviously cannot be reserved for locating and clamping purposes because locators and clamps would interfere with the cutting tools. Disappearing or retracting stops are used sometimes to establish location from surfaces to be machined.

The purpose of a clamping device is to push a workpiece firmly against its locators and hold it there securely against all cutting forces. A clamping device must apply adequate forces to meet those demands but must not distort or damage a workpiece. For instance, clamping forces A and B in Fig. 8-4 are applied to press the workpiece against the locating points. The resulting frictional forces prevent the workpiece from turning or moving axially in the locators when cutting is done. If the workpiece were a thin tube. it might be distorted. A solid bar would not be hurt. To avoid upsetting location or impairing the workpiece, clamping forces should be directed inside the areas of location through strong and rigid workpiece sections.

The part of Fig. 8-1 is to be made from a round bar that should be located from the equivalent of four points as shown in Fig. 8-4. If a number of pieces were to be made on a semiautomatic turning machine, a stop would be necessary to locate the outer end of the bar. The stop would be moved out of the way after the bar was clamped. On such a machine the tools move in definite paths, and each piece must be located with respect to them. In contrast, an end stop is not so necessary when one or a few pieces are made on an engine lathe. The tools are adjusted to each workpiece.

The surfaces and spots for locating and clamping are selected first on the workpiece. After that, a holding device is chosen to act upon the desired areas.

To Select the Tools, Machines, and Other Equipment

So far consideration has been given to what must be done to the workpiece. The next problem is to find what equipment does the job best. As a start the field can be narrowed to a few types of machines and tools designed specifically to do what is needed. For example, the decision has been made to revolve the workpiece of Fig. 8-1 while it is turned, faced, drilled, bored, reamed, chamfered, and cut off. That precludes the use of the shaper, planer, milling machine, broaching machine, and many others. The candidates left for the job are the machines in the lathe family.

The features of machines and tools that must be considered. The best machines and tools are selected for the purpose at hand from those remaining by appraising their features. The features of a machine tool that suit it for certain kinds of work are:

1. *The cost of operation.* Two machine tools may be capable of doing the same work. One is an engine lathe that can be set up easily for the job but is slow. The other is an automatic lathe that takes more time for setup but machines each piece rapidly. A special machine that can be used on only one job must be paid for by the earnings from that job.
2. *Range and capacity.* These are related to the size of the machine. On a lathe this is a matter of the diameter and length of work that can be accommodated. The size of the work is not as important for some machine tools as the area or length of cut to be made. The speeds and feeds available on a machine tool fall within a specific range. The range is broad on some machines, narrow on others. Two machines may be alike except that one has a high range of speeds, and the other a low range.
3. *Strength and power,* or ability to take the cuts demanded at economical rates.
4. *Dependability and accuracy.* Some low-priced machine tools are not intended for the most accurate work. An old outworn machine cannot be expected to perform reliably.
5. *Availability.* This is an important practical consideration in many cases where it is necessary to use the equipment on hand rather than a more ideal machine that would have to be purchased.

The features of a cutting tool that are taken into account are:

1. *The type of tool,* whether it has one or more teeth, the way it is held, etc.
2. *The form and size of tool,* whether it is a single point or or form tool, right- or left-hand, frail or rugged, etc.
3. *The shape of the tool,* as determined by its angles.
4. *The tool material.*
5. *Availability and cost.*

The important features of a holding device are:

1. *The way it locates a workpiece.* This must correspond to the method of location decided upon for the operation.
2. *The ability to clamp the work securely* without distorting it.
3. *The accuracy of location.* Various kinds of chucks, for instance, are available with different degrees of accuracy.
4. *Availability and cost.* Each machine tool is normally equipped with certain standard holding devices. If others are wanted, they must be made or purchased.

The features of measuring instruments and gages that need to be considered are:

1. *The degree of precision required.* Some measuring instruments are more accurate than others.
2. *Simplicity.* This concerns the ease and quickness with which the instrument can be used.
3. *Availability and cost.* Common measuring instruments and certain standard gages generally are on hand in most plants. Others have to be made or purchased, which means additional cost and perhaps delay.

The factors that determine what features are necessary. Equipment is selected for an operation because it has features that are best for the particular job. Certain factors must be considered to ascertain what features are desirable. The quantity of parts to be produced by an operation is one of the most important factors. The cost per piece is the total cost of conducting the operation divided by the number of pieces produced. The total cost is the sum of the direct charges for labor, power, etc., preparation and setup costs, overhead charges, and the cost of using the equipment. The

proportions of these costs differ from one operation to another.

As an example of costs, the piece of Fig. 8-1 may be machined on an engine lathe or turret lathe. The cost of the operator for either machine is \$2.00 per hour. Overhead including capital charges is \$4.00 per hour for the engine lathe and \$6.00 per hour for the turret lathe, which represents a larger investment. About 20 minutes are required to set up the engine lathe for the job, but one hour is needed for the turret lathe. The machining time is 6 minutes per piece on the engine lathe but can be reduced to 3 minutes per piece on the turret lathe. If only one piece is to be made, the cost on the engine lathe is $(1/3 + 1/10) \times 6 = \$2.60$ and on the turret lathe is $(1 + 1/20) \times 8 = \$8.40$. The engine lathe is more economical for only one piece.

Suppose that 75 pieces of Fig. 8-1 were required. The setup time remains the same for both machines. The cost on the engine lathe is $(1/3 + 75/10) \times 6/75 = \$0.63$ per piece, but on the turret lathe is $(1 + 75/20) \times 8/75 = \$0.51$ per piece. The turret lathe is more advantageous for larger quantities.

Standard basic machine tools with conventional tools generally are best for small quantities of parts. They produce at slow rates and require skilled operators but impose the least overhead and capital charges upon individual operations. They may be equipped with some special tools for somewhat larger quantities. Other machines are designed for moderate quantities of output, are fairly simple, and are arranged partly with standard tools and partly with special tools. Semiautomatic or fully automatic machine tools, often quite complicated and with highly refined tooling, are suitable for large quantities of parts. No universal distinction can be made among small, moderate, and large quantities. A situation calling for a large number of simple parts may be comparable to one for a small number of complex pieces. Each case must be decided on its own merits.

Practically all work can be done with standard cutting tools, holding devices, machine tool attachments, and measuring tools. Special tools and gages are made and used to save time, but are justified only when they can serve to produce enough pieces to justify their cost.

The machine tool, holding device, and cutting tools for an operation cannot be selected separately. Each type of machine tool makes

use of certain kinds of cutting tools and holding devices. Even among kindred machines, tooling may differ to some extent. For instance, turning is usually done with single point cutting tools alone on an engine lathe, but box tools also are commonly found on screw machines and turret lathes.

A machine tool must have the rigidity and power to drive the cutting tools selected for it at their most efficient rates. This was emphasized in recent decades by the advent of cemented carbide cutting tools. Those tools proved able to take much heavier cuts and remove metal faster than their predecessors. The machine tools that were built for the older types of tools were not capable of getting the most from the new tools, and more advanced and powerful machines were brought into being.

Other factors that must be considered to evaluate the merits of equipment are:

1. The size and shape of the workpiece
2. The size, shape, and position of the surfaces to be machined
3. The kind of material in the workpiece and its properties
4. The amount of stock to be removed
5. The dimensional accuracy and surface finish required

The size and shape of the workpiece. Obviously, a machine tool must be large enough to accommodate a workpiece that is going to be put on it. On the other hand, too large a machine is more costly than necessary. The workpiece of Fig. 8-1 could be machined on a vertical boring mill that performs about the same basic operations as a lathe. But vertical boring mills are huge machines that are built to take pieces several feet and more in diameter. A large portion of the capacity of the machine would be wasted on the part under consideration. What is more, large machines cannot be expected to have the high speeds to make them efficient for small pieces.

The smallest conventional machine tool that will handle a job properly is the best choice, but a machine for the exact size of a workpiece is seldom available. Standard lines of machine tools progress from size to size in steps. If a machine is to be used for a variety of jobs, it must be adapted to a range of sizes. The rough stock for the piece of Fig. 8-1 is 1½ in. in diameter and might be handled on a lathe with that swing. Such a lathe would be limited

to diameters up to that size and could not handle as large a variety of work as a larger machine. A 12 in. lathe might be selected in view of its capacity to handle many other jobs beside this one.

A standard holding device can be found for a workpiece of almost any shape. Such devices may be cumbersome or slow for some pieces, and consideration of special means may be warranted.

The size, shape, and position of the surfaces to be machined. The size and shape of the surfaces to be machined on a workpiece must be considered along with the over-all size of the piece. For instance, a piece longer than a lathe can be chucked with one end supported by a center rest instead of the tailstock so that it overhangs the end of the lathe. However, the distance that can be turned on the piece is limited by the travel of the carriage.

The position of a workpiece surface may favor one type of machine as against another. Consider a piece with the surface to be machined inclined at an angle with the base. A special fixture, elaborate setup, or attachment may be needed to align the surface with the cutter on a milling machine, but not on a shaper. Machine tool attachments often are helpful in otherwise difficult situations.

The sizes of some cutting tools are related to the sizes of the surfaces they machine, as indicated by the sketches of face and slab milling in Fig. 1-3. The shape of a form cutter must obviously correspond to that of the surface it is to cut.

Cutting tools are made with many shapes to reach surfaces in various positions. The outside diameter and end face of the piece of Fig. 8-1 are machined on a lathe with a right-hand rather than a left-hand tool. Surfaces in unusual positions may require special tools. An exceptionally long drill may be needed to reach a hole that lies deep within a narrow cavity in a workpiece.

Material properties. Some materials can be cut at high speeds, others at low speeds. A machine tool must have ranges of speeds and feeds suitable for the material being cut if it is to perform an operation efficiently. Large cutting forces are needed to cut hard and tough material, and the machine tool must be strong and rigid to withstand them.

Tool angles, especially rake, depend upon work material. Tool material is likewise related. High speed steel is entirely satisfactory in many cases for soft materials, but cemented carbides are pre-

ferable for moderately hard materials, and grinding wheels are necessary for very hard materials.

The amount of stock to be removed. Stock is removed most economically by taking deep cuts and heavy feeds. The machine tool selected for an operation must have sufficient power to remove the necessary stock at an economical rate and be rigid enough to withstand the cutting forces imposed.

Cutting forces and power consumption in an operation vary with the characteristics of the tools and work material, speed, feed, and depth of cut. The effects of these variables were discussed in Chapter 4. Investigations have been made to find the actual forces acting and power consumed under many conditions. The results are given in reference books and handbooks. Table IV shows average unit values of power consumed in cutting several kinds of materials. The figures apply to many ordinary operations but may not be applicable in some cases.

Table IV

Average Values of Horsepower per Cubic Inch per Minute of Stock Removal for Some Common Materials and Machining Operations.

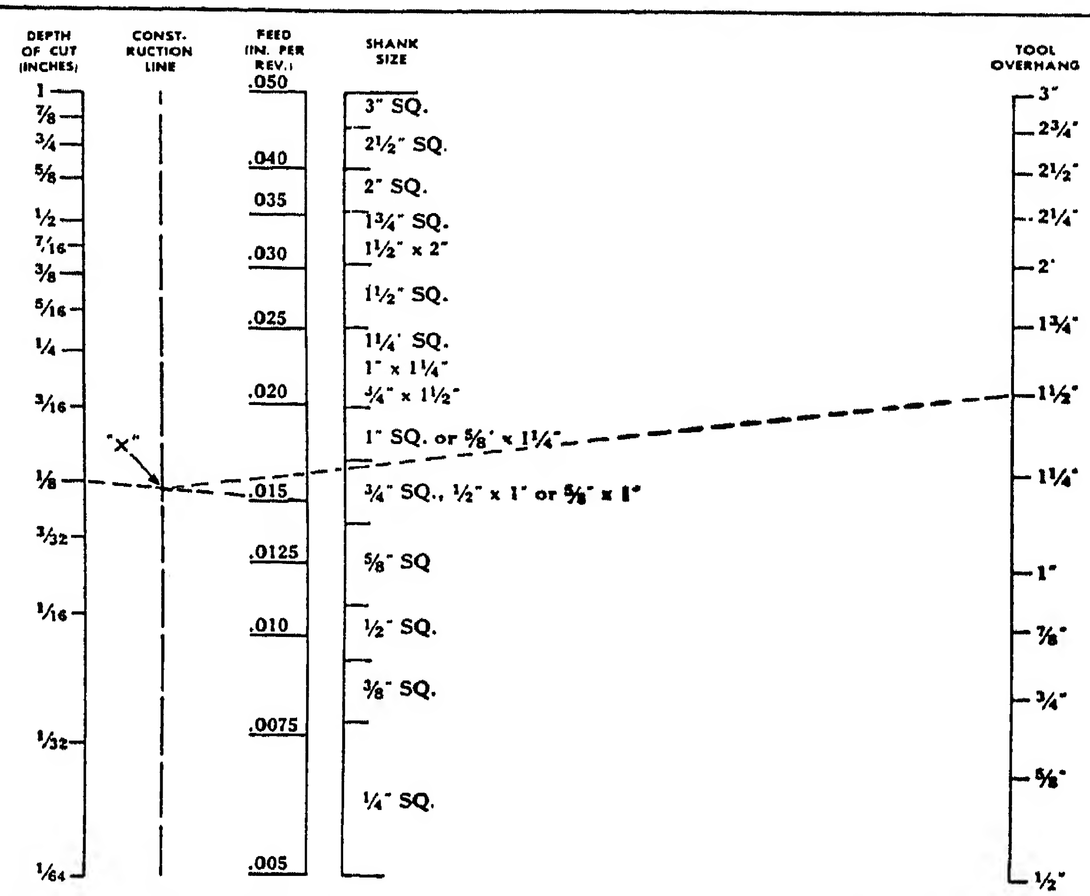
Workpiece material	*Machining operation*			
	Turning	*Drilling*	*Milling*	*Shaping*
Aluminum	0.2	0.5	0.6	0.3
Brass	0.2	0.5	0.7	0.3
Cast iron (soft)	0.3	0.6	0.8	0.4
Cast iron (hard)	0.5	0.9	1.0	0.6
Steel (soft)	1.0	1.2	1.5	1.0
Steel (medium)	1.4	1.6	1.9	1.4
Steel (hard)	2.0	2.0	2.2	2.0

The horsepower required at the tool point is the product of the unit power consumption in horsepower per cubic inch per minute times the rate of stock removal in cubic inches per minute.

The rate of stock removal with single point tools is approximated by the product of the depth of cut in inches times the feed in inches per revolution or per stroke times the cutting speed in sfpm times 12 times the number of tools.

The heaviest cut in the workpiece of Fig. 8-1 is for rough turning the outside diameter. The diameter is reduced from $1\frac{1}{2}$ in. to $1\frac{9}{32}$ in. at a feed of 0.025 in. per rev, and a speed of 71 sfpm. The rate of

stock removal is approximately $0.110 \times 0.025 \times 71 \times 12 \times 1 = 2.34$ in.^{3}/min. For soft steel, the power required at the tool point is $2.34 \times 1 = 2.34$ hp. If the machine efficiency is 75 per cent, the horsepower taken from the motor is 3.1 hp. Machine tool motors are often overloaded a little for limited periods. Consequently, a 3 hp



EXAMPLE: Assuming a feed of .015", a depth of cut of ⅛", and an overhang of 1½", you lay a straight edge between the ⅛" shown for the depth of cut on the scale at the left of the chart and the .015" point shown on the feed scale; this line crosses a construction line which lies between these two scales. Mark the point where the scale crosses the construction line. Then join this point "X" with a point on the tool overhang scale represented by the tool overhang to be used,—in this case 1½". The dashed line represents the scale. The point where this line crosses the shank size line determines the recommended shank size to use. In this case it would be ¾" square, ½ x 1, or ⅝ x 1.

Fig. 8-5. A chart for calculating shank sizes of single point tools. (Courtesy Carboloy Co., Inc.)

motor on the lathe should be satisfactory, and a 5 hp motor quite adequate.

The strength and rigidity of a cutting tool must be commensurate with the feed rate and depth of cut. The chart of Fig. 8-5, based upon the experience of a large cutting tool manufacturer, shows that large tools are needed for satisfactory performance with deep cuts and heavy feeds.

Heavy cuts favor the use of the more expensive tool materials to

remove metal fast. Multiple tooth cutters usually are able to remove stock faster than single point tools and often are more economical although their first cost is higher. For heavy cuts, consideration must be given to the space between teeth to provide room for chips.

The dimensional accuracy and surface finish required. Normal practicable tolerances are indicated in this text in the discussions of the basic machine tools and operations. Ranges of surface finishes ordinarily resulting from various methods are indicated in Fig. 2-6. Some machines, tools, and operations offer means of realizing unusually small tolerances and good finishes. Among them are those for grinding, Superfinishing, lapping, and honing.

The accuracy required in an operation is an important consideration in the selection of measuring instruments and gages because accurate dimensions have meaning only if they can be measured or gaged.

To Specify the Contents of an Operation

Instruction sheets. An instruction sheet is a form describing the planned details of an operation. Other names given the form are *operation instruction sheet, operator instruction sheet,* and *operation sheet*. Instruction sheets are written in various ways. A typical form in Fig. 8-6 describes the operation to make the screw collar of Fig. 8-1. It contains a sketch of the part with dimensions pertaining to the operation, an identification of the operation, a list of the machine and tools required, and a description of the steps that comprise the operation.

The instruction sheet of Fig. 8-6 includes setup and teardown. That is part of some forms, but not of others. Common practice is to designate the basic operations as elements of the major operation for which the sheet is written.

Determining the proper sequence of elements. The steps of an operation are not listed at random but are placed where they contribute most effectively. The operator is expected to follow the specified sequence in doing his work. Some considerations governing the arrangement of the steps are covered in the paragraphs that follow.

Some cuts are taken to prepare for others in the same operation.

17-14-1	Form 154—5M—11-34 UNIVERSITY OF ILLINOIS SHOP LABORATORIES MACHINE LABORATORY INSTRUCTION CARD	

SCREW COLLAR

OPERATION No. 1 TOTAL OPERATIONS 1

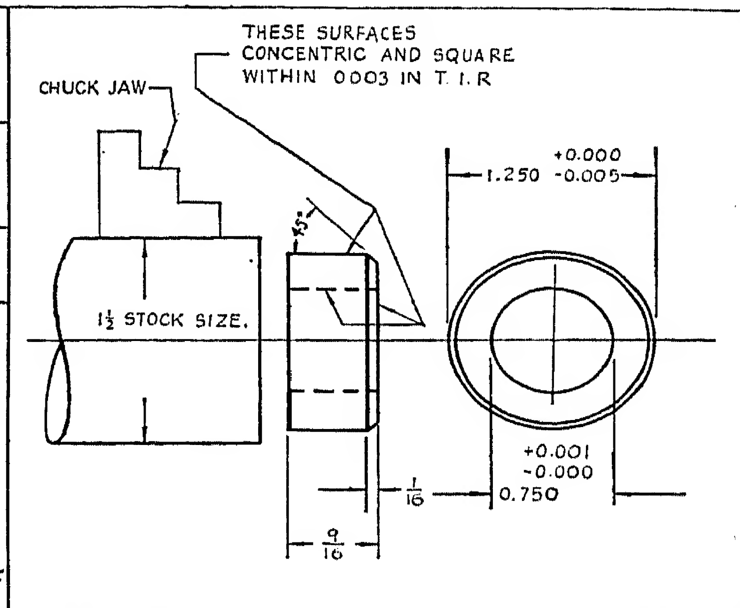
MACHINE 12"X3' ENGINE LATHE

MACHINE No. 73

TOOLS: 1-R.H. ROUND NOSE TOOL BIT
1-E1 CENTER DRILL AND CHUCK
1-5/8 DIA. T.S. TWIST DRILL
1-1/2 DIA. BORING BAR & HOLDER
1-3/4 DIA. T.S. MACHINE REAMER
1-CHAMFER TOOL BIT
1-6 IN. SCALE
1-1-2 IN. MICROMETER CALIPER

JIGS: 1-0.750-0.751 DE. PLUG GAGE
1-INSIDE MICROMETER CALIPER

FIXTURES: 1-10 IN. 3 JAW UNIVERSAL CHUCK



ITEM	OPERATION ROUTINE	SPEED S.F.P.M.	SPEED R.P.M.	FEED I.P.R.	STANDARD TIME
1	SET UP FOR CHUCK WORK			15.00	
2	CHUCK BAR				0.20
3	FACE END OF BAR. 1/32 STOCK REMOVAL	88	225	0.010	0.54
4	CENTER DRILL TO 3/16 DIA.		1200	HAND	0.35
5	DRILL 5/8 DIA. × 11/16 DEEP	90	550	0.007	0.43
6	BORE 0.742/0.747 DIA. - 2 CUTS	74	375	0.007	1.00
7	REAM 0.750/0.751 DIA.	39	200	0.031	0.35
8	ROUGH TURN 1 9/32 DIA. × 3/4 LONG.	71	180	0.025	0.68
9	FINISH TURN 1.250/1.245 DIA. × 5/8 LONG.	96	292	0.010	0.89
10	CHAMFER 1/16 × 45°	96	292	HAND	0.35
11	CUT OFF TO 9/16 THICKNESS	65	200	0.004	0.56
12	BREAK EDGES				0.25
13	INSPECT PIECE				0.40
14	REMOVE BAR AND TEAR DOWN			5.00	
	SET UP AND TEAR DOWN			20.00	
	TOTAL STANDARD TIME				6.00

Fig. 8-6. An instruction sheet for the operation of machining the screw collar of Fig. 8-1 on an engine lathe.

Roughing cuts followed by finishing are of this kind. Boring like that specified in Fig. 8-6 makes the hole true for reaming. The end of the bar is faced to prepare a good surface for the center drill and reduce runout. The preparatory steps obviously must precede those for which they prepare.

As much of the roughing as possible should be done on a workpiece before finishing is started. Heavy cuts may deflect frail pieces. Heavy cutting generates considerable heat and that causes expansion. When a large amount of stock is removed from a workpiece, internal stresses may be relieved, and distortion result. These disturbances should be allowed to run their course before the surfaces of a workpiece are finished. Where distortion from roughing is severe, the clamps that hold a workpiece may be loosened to allow the piece to assume its natural shape. The clamps then are tightened, and the surfaces finished.

The most difficult elements may be placed near the beginning of an operation. Inside surfaces usually are harder to control than outside surfaces. The hole is finished in Fig. 8-6 before the outside is turned. If the hole does not turn out right and the piece is spoiled, the outside work is not wasted.

The relationship between steps in an operation may influence the accuracy of machine settings. Suppose a 1 in. diameter and a ⅞ in. diameter are to be finished on a piece. If the larger is finished first and the smaller second, the tool is moved inward consistently in one direction and backlash is eliminated.

An arrangement that adds to the convenience of an operation makes it easier and helps minimize the time spent. On that score, a sequence that requires the least number of tool changes is preferred. If several surfaces are roughed with one tool, they should be roughed one after the other. Cuts that require the same speeds or feeds may be placed together to eliminate changes. Most pieces must be burred after machining, and judicious planning can help decrease the cost of that task. For instance, a large and small hole intersect. If the large hole is drilled first, the burr is thrown into it when the smaller one is drilled. The burr can be removed more easily from the large hole.

To Estimate Operation Time

The parts of productive time. The total time required to perform an operation may be divided into four parts. They are:

1. *Setup time.* This is the time required to prepare for the operation and may include time to get tools from the crib

THE MONARCH MACHINE TOOL COMPANY

SIDNEY, OHIO, U. S. A.

TURNING DATA SHEET No. 104

HANDLING TIME

Avg. Diam. In Inches	RECHUCK OR PLACE BETWEEN CENTERS — LENGTH IN INCHES																						
	2	4	6	8	10	12	14	16	18	20	24	28	32	36	40	44	48	52	56	60	72	84	96	108
	TIME IN MINUTES																							
3/4	.10	.10	.10	.10	.10	.12	.14	.14	.16	.16	.18	.18	.20	.20	.24	.28	.32	.36	.40	.45	.58	.71	.84	.97
1	.10	.11	.12	.15	.18	.21	24	.25	.28	.29	.32	.33	.36	.37	.42	.47	.52	.57	.62	.68	.81	.94	1.07	1.20
1¼	.14	.16	.18	.22	.26	.30	.34	.36	.40	.42	.46	.48	.52	.54	.60	.66	.72	.78	.84	.91	1.12	1.33	1.54	1.75
1½	.18	.21	.24	.29	.34	.39	.44	.47	.52	.55	.60	.63	.68	.71	.78	.85	.92	.99	1.06	1.14	1.38	1.66	1.94	2.22
2	.22	.26	.30	.36	.42	.48	.54	.58	.64	.68	.74	.78	.84	.88	.96	1.04	1.12	1.20	1.28	1.37	1.64	1.91	2.18	2.45
2¼	.26	.31	.36	.43	.50	.57	.64	.69	.76	.81	.88	.93	1.00	1.05	1.14	1.23	1.32	1.41	1.50	1.60	1.90	2.20	2.50	2.80
2½	.30	.36	.42	.50	.58	.66	.74	.80	.88	.94	1.02	1.08	1.16	1.22	1.32	1.42	1.52	1.62	1.72	1.93	2.53	3.13	3.73	4.33
2¾	.34	.41	.48	.57	.66	.75	.84	.91	1.00	1.07	1.16	1.23	1.32	1.39	1.50	1.61	1.72	1.83	1.94	2.16	2.76	3.36	3.96	4.56
3	.38	.46	.54	.64	.74	.84	.94	1.02	1.12	1.20	1.30	1.38	1.48	1.56	1.68	1.80	1.92	2.04	2.16	2.39	2.99	3.59	4.19	4.79
3½	.42	.51	.60	.71	.82	.93	1.04	1.13	1.24	1.33	1.44	1.53	1.64	1.73	1.86	1.99	2.12	2.25	2.38	2.52	3.12	3.72	4.32	4.92
4	.44	.56	.66	.78	.90	1.02	1.14	1.24	1.36	1.46	1.58	1.68	1.80	1.90	2.04	2.18	2.32	2.46	2.60	2.75	3.35	3.98	4.55	5.15
4½	.48	.61	.72	.85	.98	1.11	1.24	1.35	1.48	1.59	1.72	1.83	1.96	2.07	2.22	2.37	2.52	2.67	2.82	2.98	3.58	4.18	4.78	5.38

	PLACE AND REMOVE - HAND - ELECTRIC - AIR - 3 OR 4-JAW CHUCK—LENGTH IN INCHES																
	2	4	6	8	10	12	14	16	18	20	22	24	26	28	30	32	34
	TIME IN MINUTES																
5	.52	.66	.80	.94	1.08	1.22	1.36	1.50	1.64	1.78	1.92	2.06	2.20	2.34	2.48	2.62	2.76
5½	.56	.71	.86	1.01	1.16	1.31	1.46	1.61	1.76	1.91	2.06	2.21	2.36	2.51	2.66	2.81	2.96
6	.60	.76	.92	1.08	1.24	1.40	1.56	1.72	1.88	2.04	2.20	2.36	2.52	2.68	2.84	3.00	3.16
6½	.64	.81	.98	1.15	1.32	1.49	1.66	1.83	2.00	2.17	2.34	2.51	2.68	2.85	3.02	3.19	3.36
7	68	.86	1.04	1.22	1.40	1.58	1.76	1.94	2.12	2.30	2.48	2.66	2.84	3.02	3.20	3.38	3.56
7½	.72	.91	1.10	1.29	1.48	1.67	1.86	2.05	2.24	2.43	2.62	2.81	3.00	3.19	3.38	3.57	3.76
8	.76	.96	1.16	1.36	1.56	1.76	1.96	2.16	2.36	2.56	2.76	2.96	3.16	3.36	3.56	3.76	3.96
8½	.80	1.01	1.22	1.43	1.64	1.85	2.06	2.27	2.48	2.69	2.90	3.11	3.32	3.53	3.74	3.95	4.16
9	.84	1.06	1.28	1.50	1.72	1.94	2.16	2.38	2.60	2.82	3.04	3.26	3.48	3.70	3.92	4.14	
9½	.88	1.11	1.34	1.57	1.80	2.03	2.26	2.49	2.72	2.95	3.18	3.41	3.64	3.87	4.10		
10	.92	1.16	1.40	1.64	1.88	2.12	2.36	2.60	2.84	3.08	3.32	3.56	3.80	4.04			
10½	.96	1.21	1.46	1.71	1.96	2.21	2.46	2.71	2.96	3.21	3.46	3.71	3.96				

Use Place & Remove Time & Factor	Factor
Face Plate and Centers with Dog	.8
Face Plate and Centers with Driver	.7
Chuck and Center	1.2
Chuck and Steady Rest	1.5
Chuck and Steady Rest and Center	1.7
Air Arbor—Thread or Stud	1.0
	Min.
Pcs. On and Off Thread Arbor—Long	.50
Tighten & Loosen Headstock Center	.20
Split Bushings use in P.& R.—Add	.05
Miminum Time to P. & R. Work Between Centers	.20
Arbor Press In & Out, Add to P.& R.	.30
Pcs. In & Out of Fixtures—On or Off Table—Small	.15

11	1.00	1.26	1.52	1.78	2.04	2.30	2.56	2.82	3.06	3.34	3.60	3.86	4.12				
11½	1.04	1.31	1.58	1.85	2.12	2.39	2.66	2.93	3.18	3.47	3.74	4.01					
12	1.08	1.36	1.64	1.92	2.20	2.48	2.76	3.04	3.30	3.60	3.88						
13	1.12	1.41	1.70	1.99	2.28	2.57	2.86	3.15	3.42	3.73	4.02						
14	1.16	1.46	1.76	2.06	2.36	2.66	2.96	3.26	3.54	3.86							
15	1.20	1.51	1.82	2.13	2.44	2.75	3.06	3.37	3.66	3.99							
16	1.24	1.56	1.88	2.20	2.52	2.84	3.16	3.48	3.78	4.12							
17	1.28	1.61	1.94	2.27	2.60	2.93	3.26	3.59	3.90								
18	1.32	1.66	2.00	2.34	2.68	3.02	3.36	3.70	4.02								
19	1.36	1.71	2.06	2.41	2.76	3.11	3.46	3.81									
20	1.40	1.76	2.12	2.48	2.84	3.20	3.56	3.92									
21	1.44	1.81	2.18	2.55	2.92	3.29	3.66	4.03									
22	1.48	1.86	2.24	2.62	3.00	3.38	3.76										
23	1.52	1.91	2.30	2.69	3.08	3.47	3.86										
24	1.56	1.96	2.36	2.76	3.16	3.56	3.96										
Over	1.60	2.01	2.40	2.83	3.24	3.65	4.06										

Medium	.30
Large	.50
Extra Large	.80
Tighten Nut and Bolt	.20
Tighten Clamp 1 Nut or Bolt	.25
Tighten Clamp 2 Nut or Bolt	.40
Clamp On & Off 1 Nut or Bolt	.50
Clamp On & Off 2 Nut or Bolt	.70
Jack Moved to P. & R. Piece	.60
Clean Chips Off Job	.30
Adjust Set Screw	.10
Place & Remove Shim in Fixture	.05
Nut Arbor—Add to P. & R.	.50

Add one (1) minute for hoist on piece over 35 pounds.
(lbs.=D^2 × length × .22)—Solid piece.

Double above times for Independent 4-Jaw Chuck—Includes ordinary indicating. Tubing or like pieces—Factor .5

TOOL ADJUST—MEASURING—SPECIAL ALLOWANCES

Short Operations by Stock Diameter	¾″	1¼″	2″	3″	4″	5″	7″	10″	15″	Over
Spot Drill or Center	.10	.15	.20	.25	.30	.35	.40	.45	.50	.55
Chamfer—Bevel—Radius—Neck—Recess	.05	.10	.15	.20	.25	.30	.35	.40	.60	.80
Use these Material Factors with above operations, S A E 1020, 1112, Cast Iron, Brass - 1.00 — Machine Steel, Bronze, Tubing - 1.50 — Tool Steel - 2.00										
Measuring Times by Outside-Inside Diameters or Lengths	½″	1″	2″	3″	4″	5″	7″	10″	15″	Over
Scale Short Length by O. D.	.10	.10	.15	.15	.15	.20	.20	.25	.30	.35
Scale Long Length (over 4 feet) by O. D.	.35	.35	.40	.40	.50	.50	.60	.70	.80	.90
V-Gauge—Calipers by O. D.	.20	.25	.25	.25	.30	.30	.40	.50	.60	.70
Thread Gauge Male by I. D. Female by O. D.	.10	.15	.20	.25	.35	.45	.65			
Plug Gauge by I. D.	.05	.05	.10	.20	.30	.30	.40	.50	.60	.70
Taper Gauge Male by I. D. Female by O. D.	.20	.25	.30	.35	.40	.45	.50	.60	.70	.80
Bevel Protractor	.20	.20	.20	.30	.30	.30	.40	.50	.60	.70
Depth Micrometers by Length of Depth	.15	.20	.25	.30	.35	.40				
Micrometrs and Verniers (X2 on inside mikes)	.05	.10	.15	.20	.25	.30	.40	.50	.75	1.00
Height Gauge	.10	.10	.15	.15	.20	.20	.25	.25	.30	.30
Tool Adjust By Stock Diameters	¾″	1¼″	2″	3″	4″	5″	7″	10″	15″	Over
Hexagon Turret or Cross Slide	.10	.15	.17	.20	.25	.27	.30	.35	.40	.45
Tailstock (Hole Diameter)	.20	.30	.40	.50	.60					
Tool Adjust on Work Supported on Both Ends—By Length	6″	12″	18″	24″	30″	36″	48″	60″	72″	Over
Cross Slide	.10	.15	.20	.25	.30	.35	.40	.45	.50	.55

Form No. M-251 3M-9-45

Fig. 8-7. Standard elements of handling time for lathes. (Courtesy The Monarch Machine Tool Co.)

and do paper work as well as to arrange the tools on the machine.

2. *Man or handling time.* This is the time the operator spends in loading and unloading the work, in adjusting the machine and tools, and in making measurements during each cycle of the operation.
3. *Machine time.* This is the time during each cycle of the operation that the machine is working or the tools are cutting.
4. *Down or lost time.* This is the unavoidable time lost by the operator because of breakdowns, waiting for tools and material, etc.

Setup, man, and down time. Setup time is performed usually once for each lot of parts. It should therefore be listed separately from the other parts of the operation time. If 30 minutes are required for a setup and only ten pieces are made, an average of 3 minutes of setup time must be charged against each piece. On the other hand, if 60 pieces are made from the same setup, only ½ minute is charged per piece. Thus a prorated setup time may be very misleading because it depends so much on lot size.

Both setup and man time are estimated from previous performance on similar operations. All work on a particular type of machine tool consists of a limited number of elements, selected and arranged to suit each operation. These elements may be standardized, measured, and recorded. That is the essence of the subject of Time Study, a large field in itself. Space does not permit a detailed treatment of that subject in this text. An example of handling time standards for lathes is given in Fig. 8-7.

The actual amount of down time that will occur in a specific operation can scarcely be predicted. Some operations will run smoothly, others will be beset by troubles. The best estimate that can be made is based upon the average amount currently lost in the plant.

Machine time. Machine time can be calculated for cuts where the feed of the tool takes place at a definite rate. The basic relationship is that the time for a cut in minutes is equal to the quotient of the distance the tool is fed in inches divided by the feed expressed in inches per minute

The distance that a tool is fed to make a cut is the sum of the distance the tool travels while cutting to full depth, its approach distance, and its overtravel. The *approach* is the distance a tool is fed from the time it touches the workpiece until it is cutting fully. Approach distance for a drill is the length of its point, which is about one-fourth the diameter of a standard drill. The approach of most single point tools is negligible. *Overtravel* is the distance the tool is fed while it is not cutting. It is the space over which the tool idles before it enters and after it leaves the cut. Overtravel usually is from 1/32 to 1/4 in.

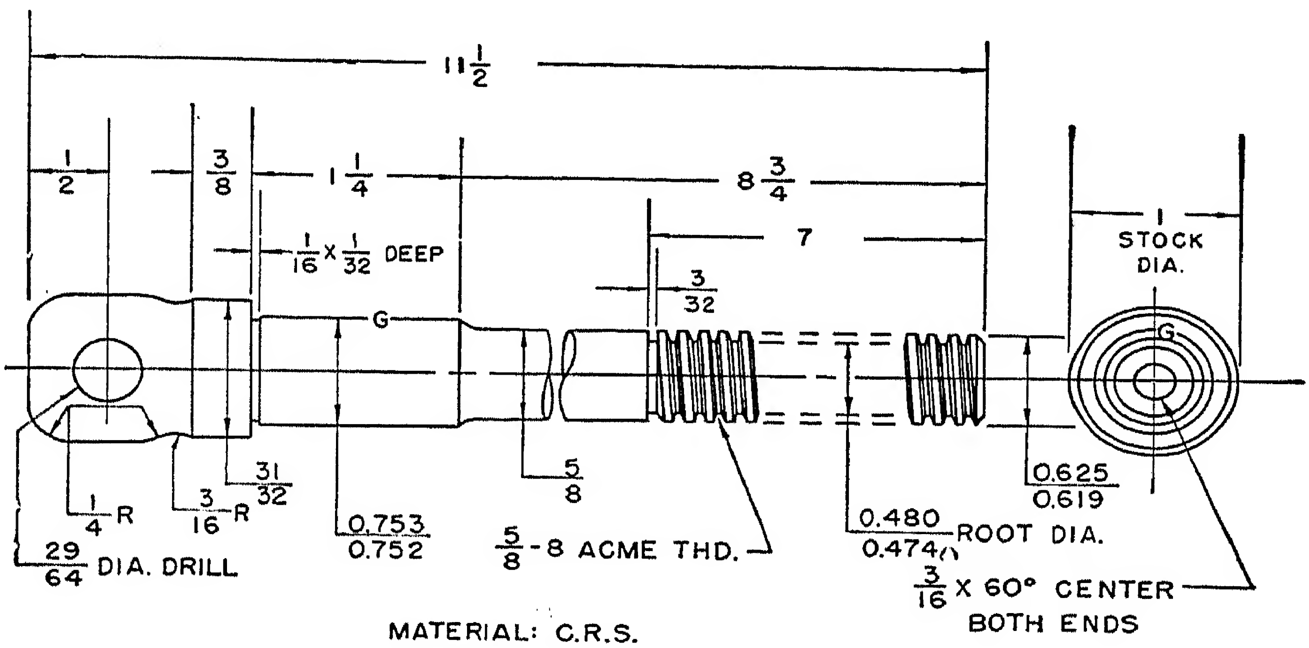


Fig. 8-8. A vise screw.

Feed on a lathe is specified in inches per revolution. The feed rate in inches per minute is found by multiplying the feed rate in inches per revolution by the number of revolutions per minute of the work or tool.

Procedure of estimating. A study of several of the time values on the operation sheet of Fig. 8-6 will illustrate the use of the principles of operation time estimating. The time for the element of "chuck bar" is taken directly from Fig. 8-7. The basic operation of "face end of bar, 1/32 stock removal" contains two elements. One is a handling time element of "tool adjust" with a value of 0.15 min from Fig. 8-7. The other is a machine time element calculated in the following manner:

The full diameter on the end of the rough bar is faced. The length

of cut is ¾ in., approach and overtravel are assumed to amount to ⅛ in., and the total tool travel is ⅞ in. For 0.010 in. per rev feed and 225 rpm, the feed rate is 2.25 in. per min. The time for the cut is ⅞ ÷ 2.25 = 0.39 min. The total time for the basic operation is 0.15 + 0.39 = 0.54 min.

Item No. 4 in Fig. 8-6 specifies, "Center drill to 3/16 diam." Feed is by hand and machine time is not calculated. Instead, from Fig. 8-7, under the heading "Short Operations by Stock Diameter—1¼ in." a value is found for "spot drill or center" of 0.15 min. To that is added 0.20 min for "tool adjust," making a total of 0.35 min.

The reamer is fed by hand to accomplish item 7, "Ream 0.750/0.751 diam," but a feed rate of 0.031 in. per rev is assumed. This is equivalent to a feed rate of 6.2 in. per min. With 1/16 in. overtravel, the length of tool travel is ⅝ in., and the machine time is 0.625 ÷ 6.2 = 0.10 min. Tool adjust from tailstock is 0.20 min, and the time to check a ¾ in. diameter hole with a plug gage is 0.05 min according to Fig. 8-7. Then the total time for item 7 is 0.10 + 0.20 + 0.05 = 0.35 min.

Process Planning

Route sheets. Single machining operations have been considered so far. They are the components of all metal machining processes. However more operations than just one generally are needed to make a part. A process is defined by a list of its component operations. A route sheet is a form that describes the operations performed to make a particular part. Other names given to that form are *operation sheet, process sheet,* and *planning operation sheet.* The purpose of a route sheet is to tell what must be done to make a part and designate the equipment to be used.

A simple route sheet is shown in Fig. 8-9 for the vise screw of Fig. 8-8. It has a heading that specifies the part, quantity, and raw material. Four columns are provided for operation numbers, operation descriptions, machine descriptions, and tools. The operations are numbered to designate their sequence. The description of each major operation includes a list of its basic operations. The machine and chief tools are specified for each operation. Sometimes only special tools like jigs, fixtures, and dies are listed. Gages also may be

ROUTE SHEET

NAME OF PART VISE SCREW PART NO. 17-4

QUANTITY 10 MATERIAL C.R.S. ROUGH SIZE 1 DIA. X $11\frac{5}{8}$

OP. NO.	OP. DESCRIPTION	MACHINE	TOOLS
1	CUT OFF TO $11\frac{5}{8}$ LONG	POWER HACKSAW	
2	CENTER BOTH ENDS	CENTERING MACHINE	1-E-1 CENTER DRILL
3	BETWEEN CENTERS, FACE ONE END TO CLEAN UP FACE OTHER END TO $11\frac{1}{2}$ O.A. TURN 0.625/0.619 THD. O.D. X 7 FROM END TURN $\frac{5}{8}$ DIA. TO $8\frac{3}{4}$ FROM END TURN 0.765/0.760 DIA. X $1\frac{1}{4}$ LONG TURN $\frac{21}{32}$ DIA. X $\frac{3}{8}$ LONG FORM $\frac{1}{16}$ WIDE X $\frac{1}{32}$ DEEP GROOVE AND FACE SHOULDER FORM $\frac{3}{32}$ WIDE GROOVE TO ROOT DIA. OF THD.-7 FROM END BREAK EDGES	10 IN. ENGINE LATHE	1-R.H. ROUND NOSE TURNING TOOL 1- 1/16 WIDE GROOVING TOOL 1-R.H. SHARP POINT FACING TOOL
4	BETWEEN CENTERS FORM $\frac{1}{4}$ R. ON END FORM $\frac{1}{4}$ R. AND $\frac{3}{16}$ R. BLEND RADII	10 IN. ENGINE LATHE	1- $\frac{1}{4}$ R. FORM TOOL 1- $\frac{1}{4}$ R. AND $\frac{3}{16}$ R. FORM TOOL
5	BETWEEN CENTERS CHASE $\frac{5}{8}$-8 ACME THD.	10 IN. ENGINE LATHE	1-8 P. ACME THD. TOOL 1- $\frac{5}{8}$-8 ACME THD. GAGE
6	LAYOUT POSITION OF AND DRILL $\frac{29}{64}$ DIA HOLE	15 IN. DRILL PRESS	1-VEE BLOCK 1- $\frac{29}{64}$ T.S. DRILL
7	BETWEEN CENTERS GRIND 0.753/0.752 DIA.	6 X 18 PL. CYL. GR.	
8	INSPECT	BENCH	

Fig. 8-9. A route sheet for the process of making 10 screws of Fig. 8-8.

included. Some route sheets contain more information such as speed and feed specifications, standard times, etc. Other route sheets are more brief.

Principles of process planning. A route sheet reflects a plan

made for a process. The procedure for planning a process is similar to that for planning an operation. First, what is to be done is determined. Second, the ways of treating the workpiece are visualized to meet the specifications and requirements. The treatment to be given the workpiece determines the basic operations. The workpiece is studied to find the surfaces best suited for locating, measuring, and gaging. These are called *critical areas.* The 60° center holes and ends of the workpiece are the critical areas of the part of Fig. 8-8.

Determination of the contents and sequence of the major operations is next. Related basic operations are naturally grouped together. Thus, basic operations that require turning can be put in one group, milling in another group, etc. One or more major operations are formed from the items in each group. On the route sheet of Fig. 8-9, the lathe work is divided among operations 3, 4, and 5. The operations should be arranged to utilize the equipment and operator most efficiently in each case.

The first machining operations should be devoted primarily to machining the critical areas to make them available for locating purposes in subsequent operations. That is done in operation 2 and in the beginning of operation 3 of the routing of Fig. 8-9. Other basic operations should be done along with those for the critical areas if feasible. Normally, the exacting finishing operations are put at the end of the process.

The selection of equipment and the sequence of steps within the major operations are based upon the considerations previously discussed for single operations.

Safety Planning and Practice

Each year injuries exact a deplorable toll from United States industry. In round numbers this amounts to 2,000,000 occupational injuries (including 16,000 fatalities and 90,000 permanent disabilities) costing \$4½ billion, as pointed out by the President's Conference on Industrial Safety in 1949. In addition, millions of noninjury accidents raise the total cost to an astronomical figure. In general, the three causes of accidents and injuries are *unsafe conditions, unsafe acts,* and *unsafe attitudes.*

Providing safe equipment. A vital factor in the elimination of unsafe conditions is the safe design, control, and operation of machine tools. Engineers have the responsibility of providing and maintaining safe equipment. Safety is provided in mechanical operations by:

1. *Safeguarding the point of operation.* If possible, provision should be made so that the operator does not need to put his hands in the area where the tools act. For that purpose, mechanical loading devices often are justified for high production operations. Guards or cages are commonly built around the danger zone. For low production, equipment may have to be arranged so that the operator must reach into the danger zone when the tools are not acting. To reduce the danger, provision may be made so that the machine cannot be started until both hands are withdrawn. One way is to require the operator to depress two buttons or levers at the same time, one with each hand.
2. *Safeguarding controls and machine mechanisms.* Starting levers may be covered by guards so that they are not tripped by falling objects. Gear case covers or working area guards may be interlocked with the power source of the machine. That is done in such a way that the machine will not start when the cover or guards are off.
3. *Guarding moving parts and power transmissions.* Most present-day machines have all shafts, belts, gears, etc. completely covered.

Eliminating unsafe acts and practices. Unfortunately, all the precautions that have been devised have been unable to eliminate either accidents or injuries. Authorities agree that most injuries result from human failures not mechanical inadequacy. The great bulk of accidents can be traced to human lapses, to the failure of individuals to think and act safely. A person must think and look ahead to perceive and avoid hazards. An engineer must cultivate this habit and instill it in others under his guidance. The best time for him to start is when he is a student.

Safety results largely from common sense and judgment. Most accidents result from acts or omissions that never should occur. A

safety-minded person is aware of the basic nature of such causes and avoids them. The practice of safety can be summarized in the following basic rules:

1. Make full use of mechanical safeguards.
2. Make full use of personal protective equipment.
3. Avoid unsure situations.
4. Care for and use hand tools correctly.
5. Dress safely.
6. Be a good housekeeper.
7. Avoid horseplay.

The significance of these rules is explained below.

Elaborate mechanical safeguards are of little avail if they are neglected when the equipment is operated. If guards are left off gear cases, belts, etc., they are useless. A machine should never be started until it is certain all safety precautions have been taken. No adjustments, repairs, or oiling should be done until a machine is fully stopped.

Many accidents can be averted by making full use of personal protective equipment. Flying chips, sparks, tool fragments, and drops of molten metal cannot be eliminated whenever metal work is done but the serious eye damage they may cause can be prevented by goggles, shields, and helmets. The need for such equipment should never be forgotten, even for the smallest job. Even the chips from ordinary turning operations have caused many eye injuries. In some plants goggles must be worn by all workers, and discharge is the penalty for failure to do so. Other personal equipment includes helmets for welding and rubber gloves and aprons to protect against corrosive chemicals.

In spite of all mechanical and personal safeguarding, situations always remain that are potentially dangerous. A dog or chuck jaws on a lathe furnish an example. A work driver may be guarded by a cover for a long run setup, but adequate mechanical guards are not feasible where jobs are changed frequently. The operator must be careful to keep his hands away from the dog or chuck jaws. A wrench should never be left in a chuck. Chips are sharp and hot. If they need to be cleared away, they should be moved by a hook or rod, not by bare hands. Sharp edges and burrs should be broken from workpieces so that they do not cut someone's hands. These

and many other similar situations should be given due respect. If a cut is received, it should be given first aid treatment, no matter how small it is. The danger of infection is ever present.

Machines are not the only causes of injuries. Hand tools can be dangerous if they are defective, used for unsuitable purposes, or used improperly. They cause a large proportion of the injuries in industry. Hand tools should be well made of the best materials. Poor tools become dull, broken, or cracked quickly. Pieces are liable to fly off a chisel with a mushroomed end. A hammer with a loose head or cracked handle may come apart at any time. Constant maintenance of hand tools is essential to safety.

Even good tools may be dangerous if used incorrectly. A screw driver used as a chisel or pry bar may become knicked, split, or bent and slip out of the slot when driving a screw. The tang of a file should always be covered by a handle. Otherwise the relatively sharp point of the tang may pierce the hand or wrist if the file slips or sticks. A screw driver, file, or knife should not be used on a piece held in the hand if there is any danger whatever of the tool's slipping and cutting the hand. A cold chisel should be held to avoid striking the hand with the hammer. The safest way to do this is to grasp the chisel with the thumb and first two fingers from below. A wrench that is too large or sprung is likely to slip off a nut. If the operator is pushing on the wrench, he may be lucky to escape with only skinned knuckles.

A caption on a cartoon of a person unsafely dressed read, "An accident going somewhere to happen." Loose or torn clothing that may get caught in whirling machinery is entirely out of place in a machine shop. Clothing should be comfortable but not flowing. Sleeves should be rolled to or cut off at the elbow. A tie is best left in the locker room. Rings, bracelets, wrist watches, and similar jewelry are easily caught in moving machinery. Such accidents have cost many severed fingers or broken wrists. The hair should be covered by a cap unless quite short.

Poor housekeeping in a shop is marked by many hazards. Dirty, greasy, oily, uneven, or broken floors are an invitation to a slip, fall, and injury. Material and tools stacked or placed haphazardly hinder traffic and may even fall on someone. Clear and well-marked aisles should be provided through the shop out of the way of moving machinery. Dirty and oily rags are fire hazards and should be

placed in closed metal containers. Observance of the motto "a place for everything and everything in its place" promotes safe housekeeping.

Practical jokes, roughhousing, or horseplay have led to many injuries and not a few fatalities in machine shops. A man working near a dangerous spot may be forced into it if startled. Too often, what starts in fun gets suddenly out of hand in the grip of a swift mechanism and ends in disaster.

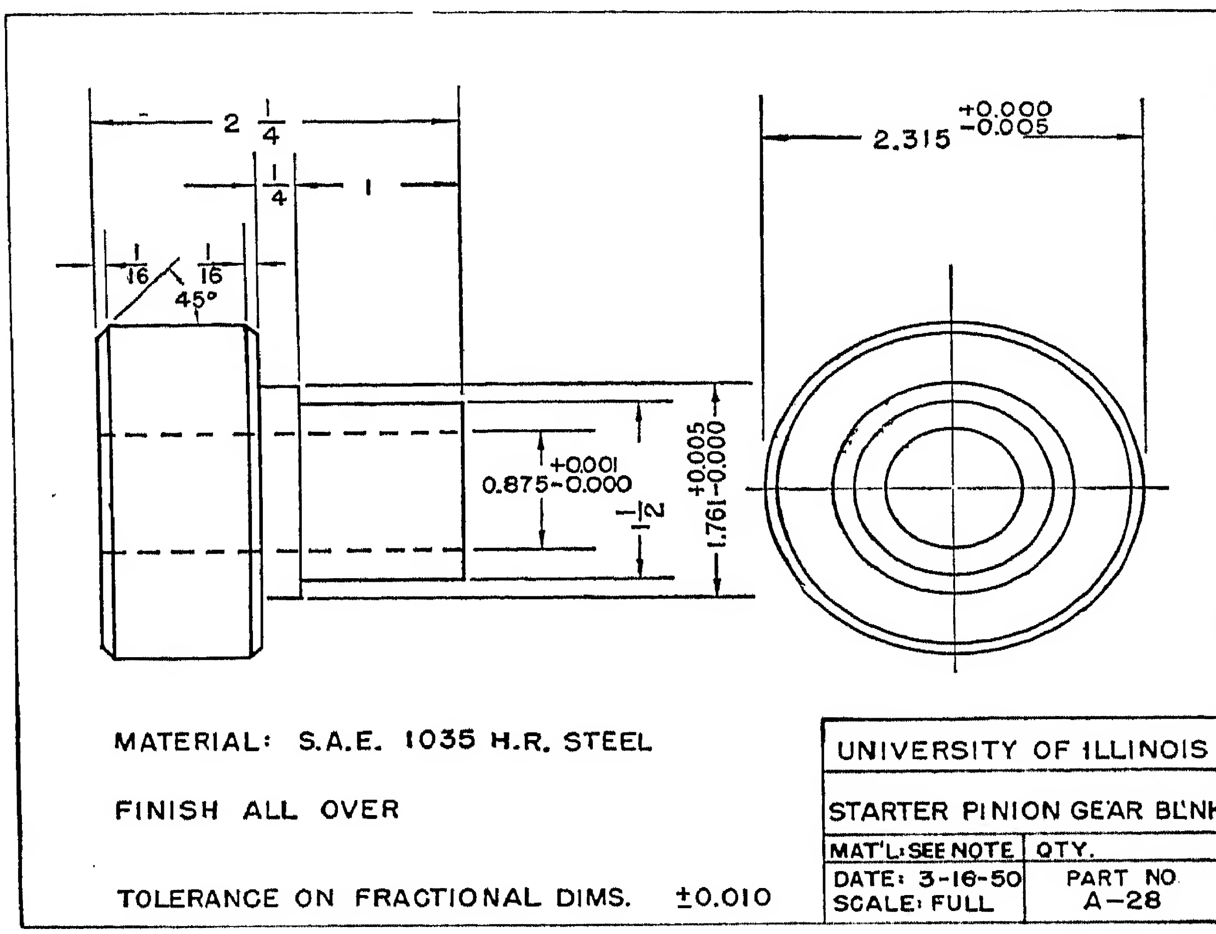


Fig. 8-10. A starter pinion gear blank.

Problems

1. Prepare an instruction sheet for machining 5 pieces like the one shown in Fig. 8–10. Specify the machine, tools, measuring instruments, and sequence of basic operations required. Select suitable speeds and feeds and estimate the time for one cycle of the operation. The blanks are to be made from a 2½ in. diameter bar.
2. Five pieces like the one shown in Fig. 8–10 are to be machined from forgings having ⅛ in. of stock on all external surfaces and no hole.
   (a) Write a route sheet showing the operations required to make the pieces to the dimensions specified.

(b) Prepare an instruction sheet for the first lathe operation.
(c) Prepare an instruction sheet for the second lathe operation.

3. Prepare a complete instruction sheet for the second operation on the route sheet of Fig. 8–9.
4. Prepare a complete instruction sheet for the third operation on the route sheet of Fig. 8–9.
5. Prepare a complete instruction sheet for the fourth operation on the route sheet of Fig. 8–9.
6. Prepare a complete instruction sheet for the fifth operation on the route sheet of Fig. 8–9.

References

Alford, L. P. and Bangs, John R., *Production Handbook.* New York: The Ronald Press Co., 1947.

American Society of Tool Engineers, *Tool Engineers Handbook.* New York: McGraw-Hill Book Co., Inc., 1949.

Doyle, Lawrence E., *Tool Engineering: Analysis and Procedure.* New York: Prentice-Hall, Inc., 1950.

Chapter 9

SHAPERS

SHAPERS ARE PRIMARILY INTENDED FOR machining flat surfaces but can be arranged for machining many kinds of curved, odd, or irregular surfaces also. Both internal and external surfaces can be shaped, in horizontal, vertical, or inclined planes, one at a time or several together.

Features of shapers. In shaping, a single point tool moves at cutting speed across the surface being machined. The tool travels in a straight line to cut a plane surface but may be guided in curved or irregular paths for other kinds of surfaces. The tool cuts while traveling in one direction only and then is returned to its starting position to begin another cut. The work is fed intermittently to present fresh material to the tool at the beginning of each cutting stroke.

Because the tool cuts only part of the time, shaping is slow as compared with other methods that do much the same work and cut continuously. Consequently, shaping is not generally considered a production process. A shaper is easy to set up and can be changed readily from one job to another. Because of their flexibility and versatility, shapers are widely used in toolrooms and jobbing shops where few identical pieces are produced.

Sizes and Types of Shapers

The size of a shaper designates its longest nominal cutting stroke. Thus, a 24 in. shaper has ram travel enough to drive a tool across a 24 in. long surface, sometimes a little more. On most shapers, the table can be fed crosswise a distance at least as large as the stroke. That means a 20 in. shaper usually is capable of machining a plane surface 20 in. by 20 in. square.

Horizontal shapers. A horizontal shaper has a ram that reciprocates in a horizontal plane and an adjustable table for the work. Horizontal shapers range from bench models with strokes of 7 or 8 in. to heavy duty models with strokes of 36 in. on push cut shapers and 72 in. on draw cut shapers. The capacity of a shaper is limited by the feasible size of the overhanging ram.

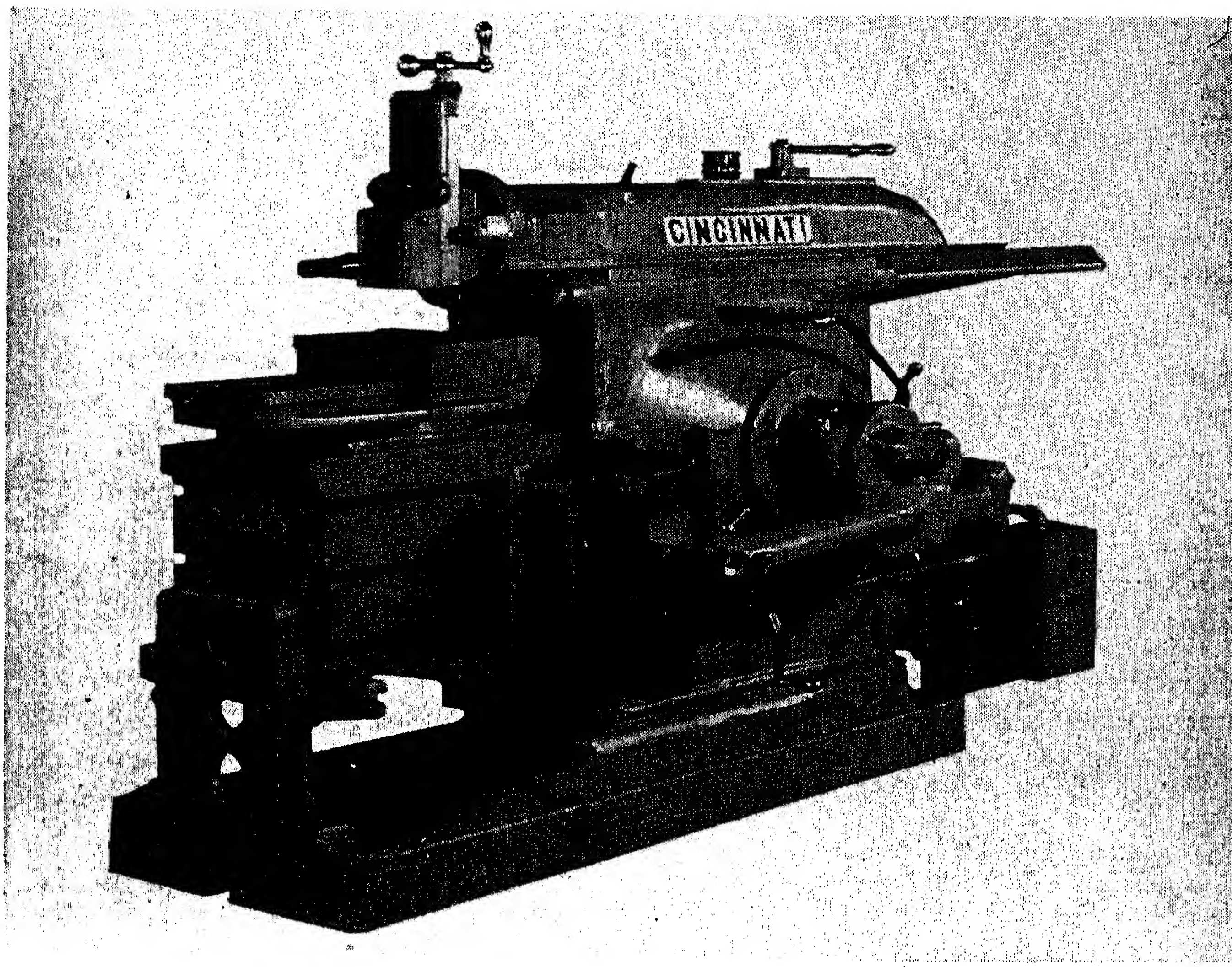


Fig. 9-1. A 24 in. standard horizontal crank operated push cut shaper. (Courtesy The Cincinnati Shaper Co.)

A *plain or utility shaper* is a light shaper with a plain adjustable table. *Standard, heavy duty,* or *production shapers* are heavy and rugged and are characterized by an adjustable table supported at its front end, as shown in Fig. 9-1.

A *universal shaper,* exemplified in Fig. 9-2, has a table that can be swiveled around two horizontal axes and also adjusted horizontally and vertically.

Several kinds of drives are built for shaper rams, and the name given to a shaper frequently denotes the kind of drive it has. One kind is through a *mechanical crank*, another is *hydraulic*. A less common kind is the *gear drive*, in which a rack on the ram is driven by a spur gear. A shaper with that drive is called a *geared shaper*. The shaper of Fig. 9-3 has a gear drive.

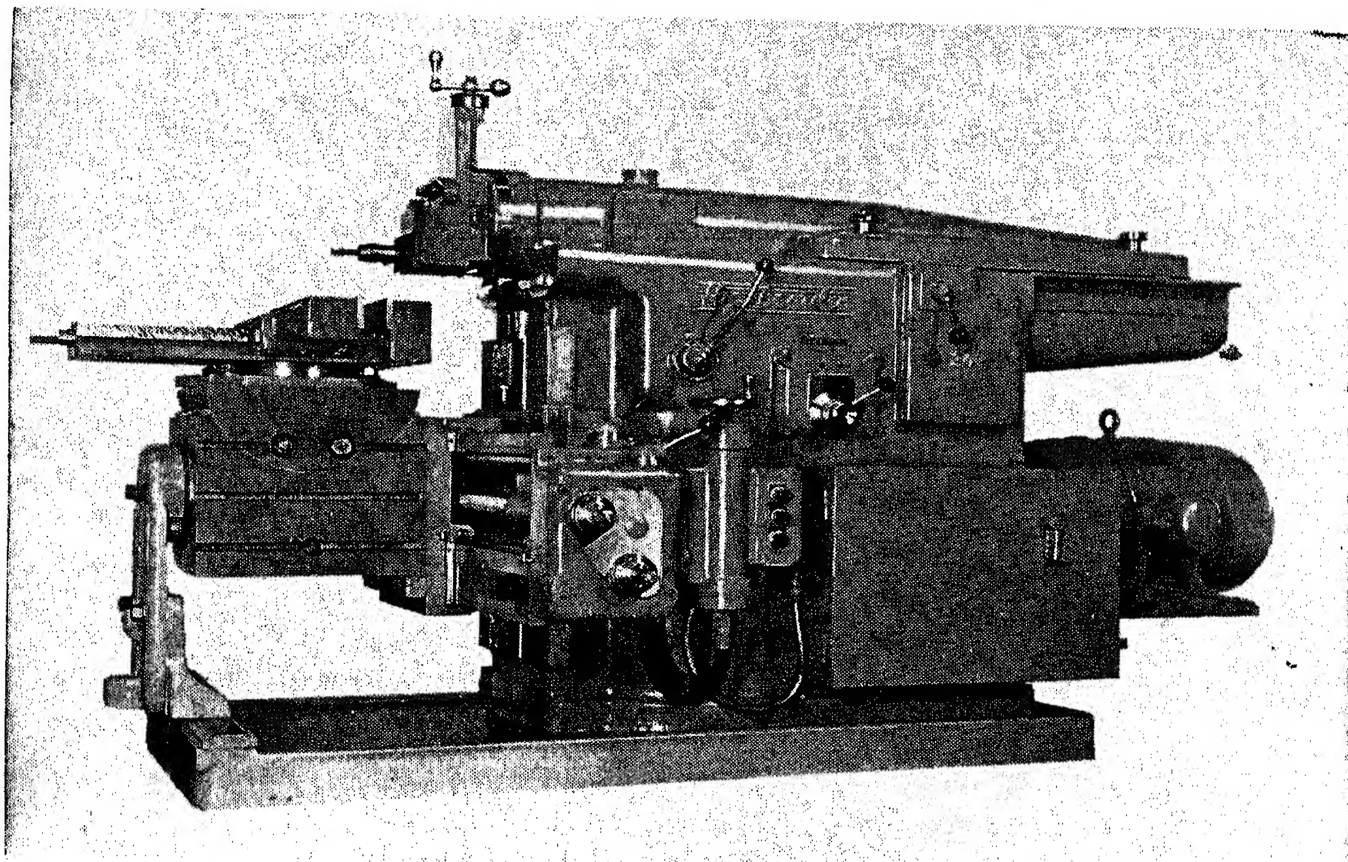


Fig. 9-2. A 24 in. universal high speed hydraulic drive shaper. (Courtesy Rockford Machine Tool Co.)

A *draw cut shaper* cuts inward toward the column instead of outward like other horizontal shapers. The inward direction of cut stablizes the table and ram which helps heavy cuts. The draw cut shaper of Fig. 9-3 has its ram backed by a heavy overarm. The table is supported by four screws. Heavy cuts and a variety of form cuts can be taken with less vibration and chatter than is experienced on standard shapers. Chips are thrown away from the operator, contributing to safety. The operator can readily judge and control the cut that starts on the outside. Machines of this type can be made

with exceptionally long strokes. They are found in railroad shops, heavy equipment plants, and die shops.

Other types of shapers exist besides those described, but they serve particular purposes and are not common. One of these is a *traveling head shaper* that has a box-shaped bed with two tables.

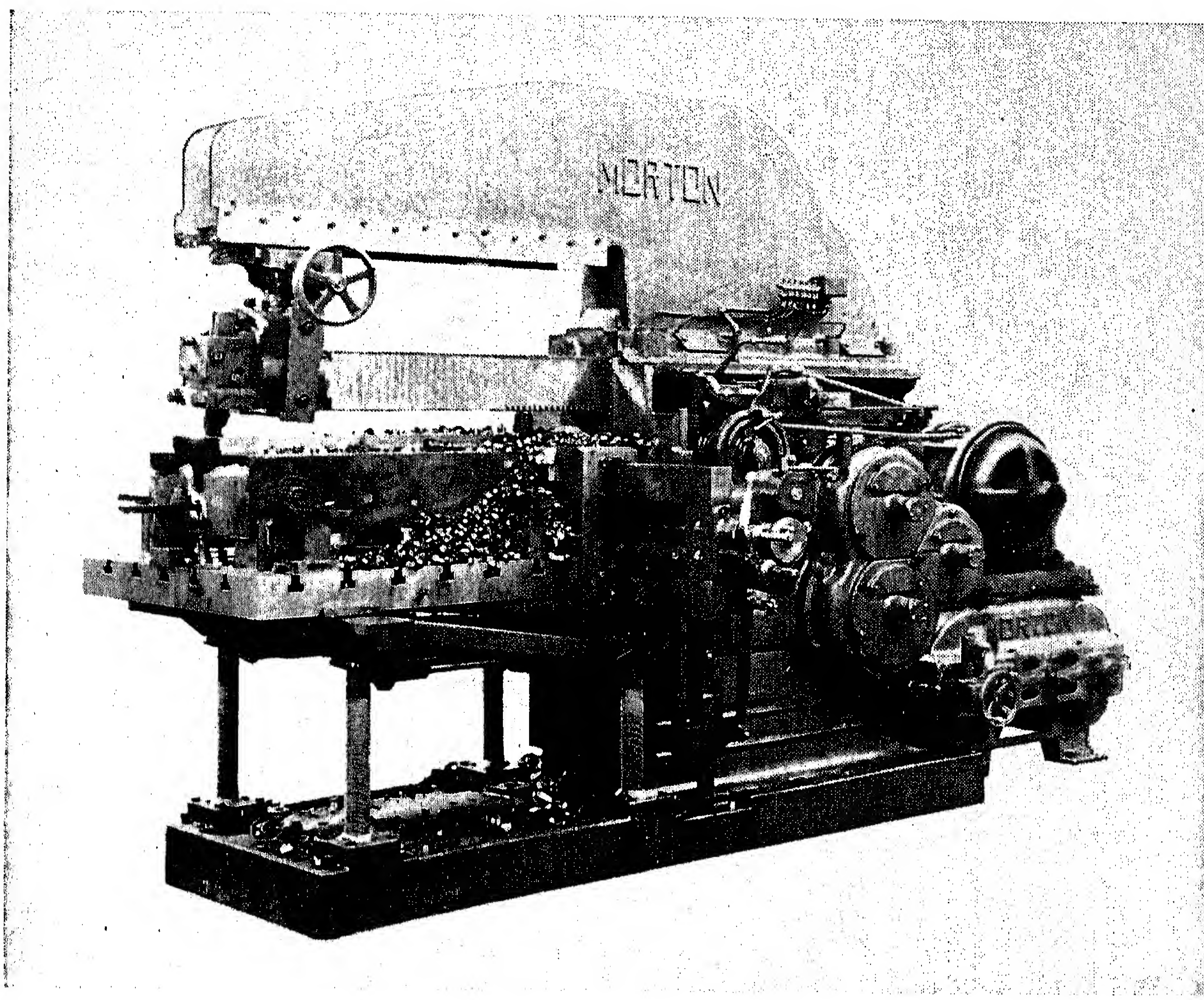


Fig. 9-3. A 42 in. heavy duty draw cut shaper machining a 20 in. by 35 in. by 12 in. die block. (Courtesy Morton Manufacturing Co.)

The ram is carried by a saddle that moves crosswise to position the ram with respect to each table in turn. A *double head shaper* has two rams to work on two workpieces at a time mounted on a table.

Vertical shapers. A vertical shaper or *slotter* has a vertical ram and normally a rotary table as illustrated in Figs. 9-4 and 9-5. The ram travels only in a vertical path on some machines but is inclinable up to 10° from the vertical on others. Some are crank operated,

others hydraulically driven. The rotary table can be indexed accurately or rotated continuously by hand or power. It is carried on a saddle that slides on the bed with horizontal adjustments and feeds in two directions. These machines are rated from 6 to 20 in., with strokes on some up to 36 in.

The horizontal table of the vertical shaper is easy to load. A bulky or heavy piece can be set on the table by a crane and slid

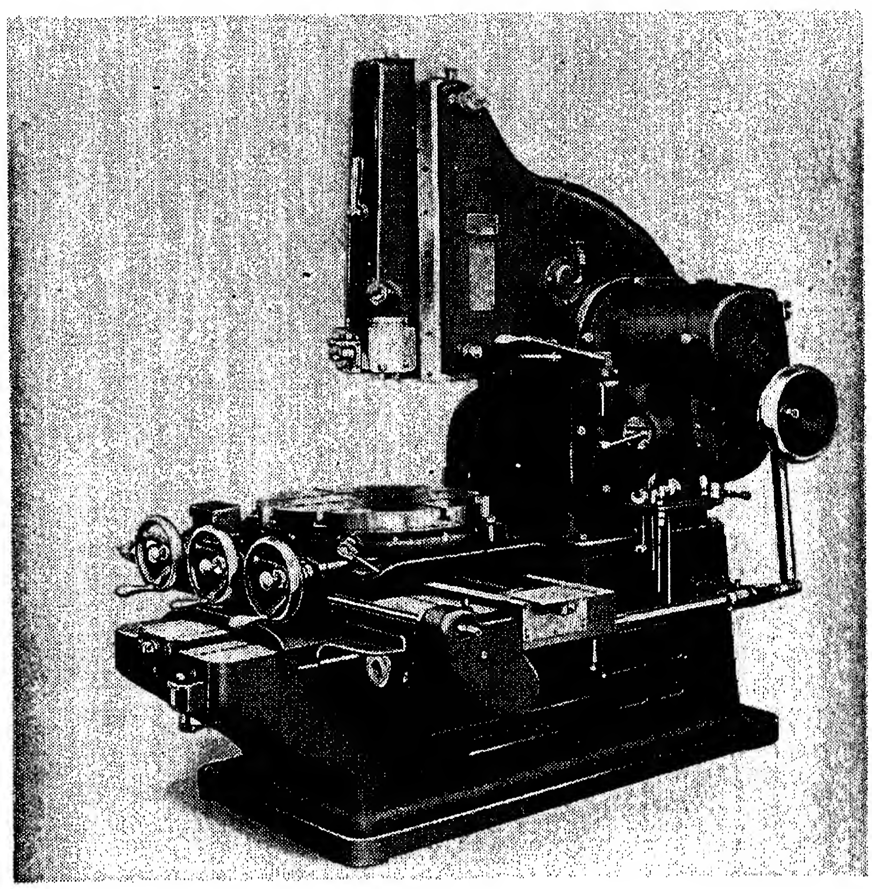


Fig. 9-4. A 12 in. crank operated vertical shaper. (Pratt and Whitney Photo from Pratt and Whitney Division Niles-Bement-Pond Co., W. Hartford, Conn.)

about until located as desired. The circular plus the two straight line motions of the table permit easy machining of circular, convex, concave, and irregularly shaped surfaces. Slots or grooves can be accurately spaced around a piece. Large internal and external gears are often finished on the vertical shaper.

The vertical design requires a minimum of floor space and makes possible a convenient grouping of controls. The operator can easily observe and control the work being done. The tool enters the work

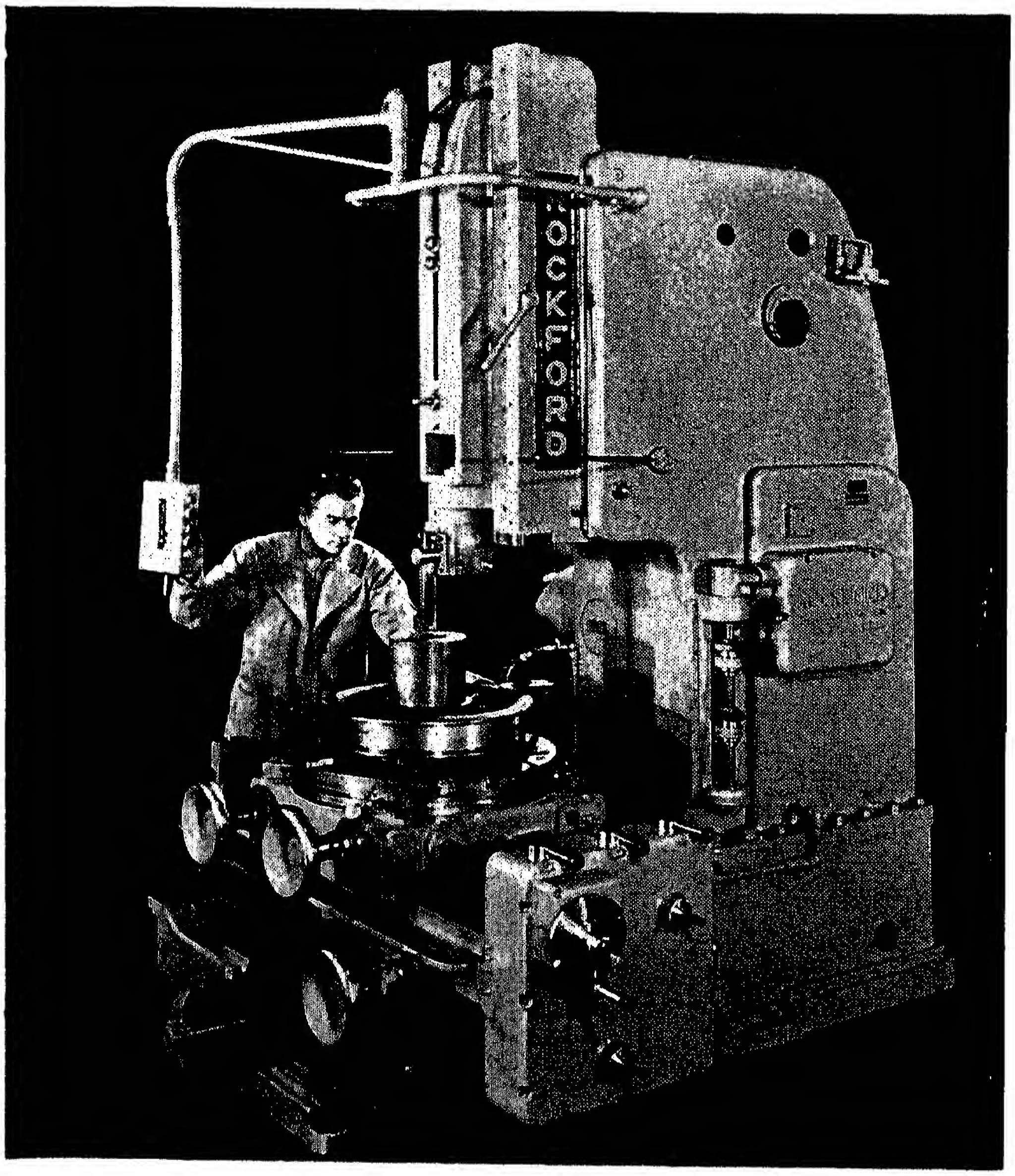


Fig. 9-5. A 20 in. hydraulic vertical shaper or slotter machining an internal keyway. (Courtesy Rockford Machine Tool Co.)

on the guide line on top of the work and can be held to the line with little difficulty.

Gear shapers are special purpose shapers for cutting gear teeth and are described in Chapter 20.

Elements of Shapers and Their Functions

Base and column. The base or bed of a shaper is the foundation for the other main units, particularly the column that contains the drive mechanism and carries the ram. These units are massive and rigid to absorb vibrations and withstand cutting and driving forces.

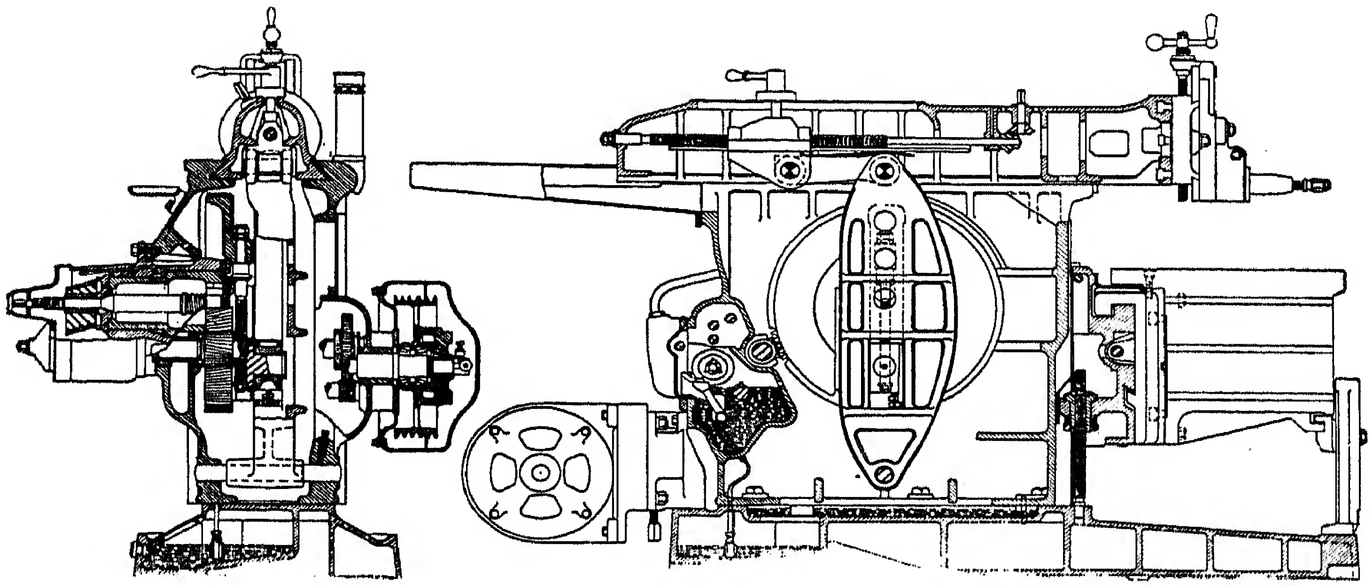


Fig. 9-6. Cross-section views of a crank driven shaper. (Courtesy The Cincinnati Shaper Co.)

Crank drive. The mechanism of a typical crank-operated shaper is shown in Fig. 9-6. Power goes from the motor to a pulley and passes through a clutch and gear transmission to the pinion that drives the large bull gear in the middle of the column. The machine is started and stopped by the clutch operated by a lever on the side of the column. Speed is varied by shifting gears in the transmission.

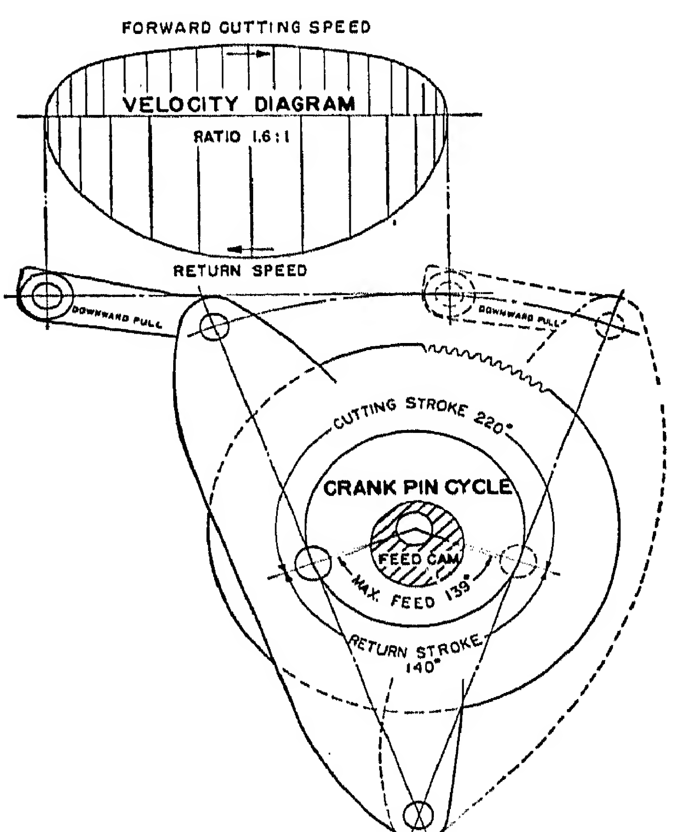


Fig. 9-7. A diagram of the crank mechanism of a shaper. (Courtesy The Cincinnati Shaper Co.)

A diagram of the crank mechanism of a shaper is shown in Fig. 9-7. The bull gear rotates clockwise at a uniform rate. A block on a crank pin on the bull gear slides in a slot of the crank that is pivoted at the bottom and connected at the top to the ram through a link. The direction of movement of the crank changes when the centerline of the crank is tangent to the circular path of the crankpin on the bull gear. When the ram moves forward on the cutting stroke, the driving pin travels

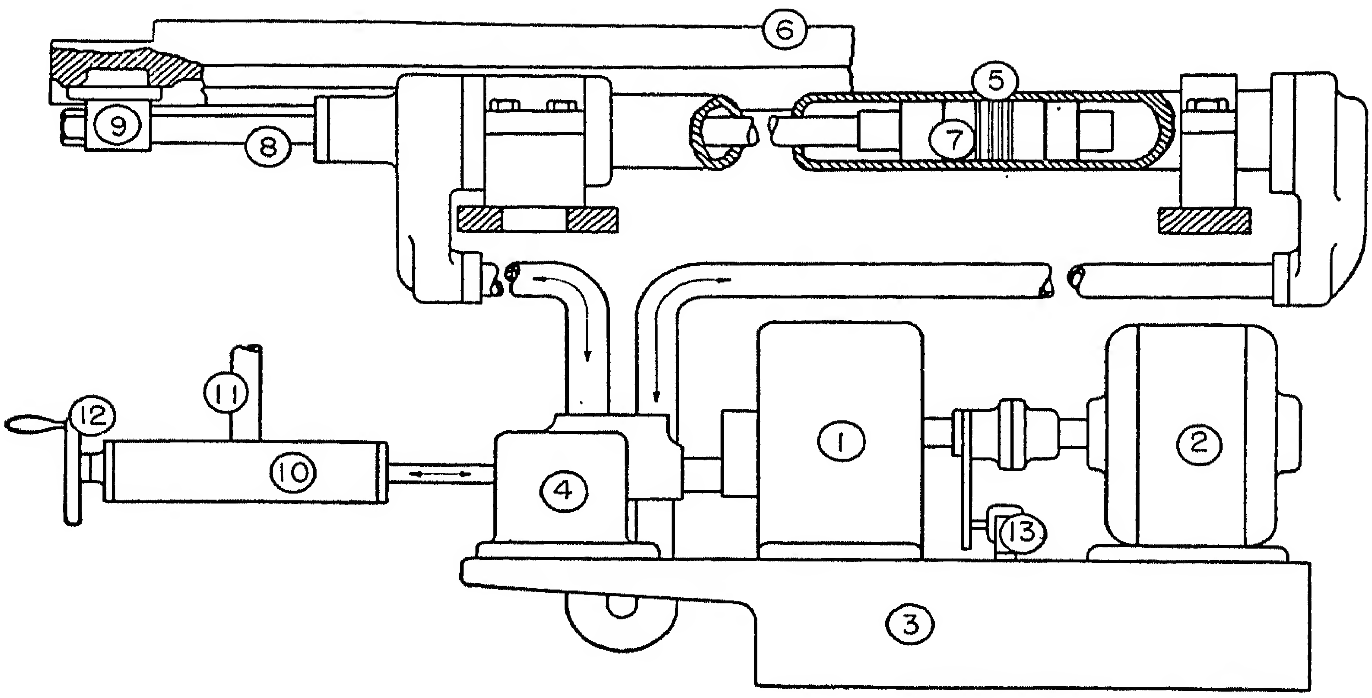


Figure. 9-8. A diagram of a hydraulic circuit for shapers and planers. (Courtesy Rockford Machine Tool Co.)

through an arc of 220°. On the return stroke, the pin travels through only 140° of arc. Thus the ram travels much faster and takes less time on the return stroke than on the forward or cutting stroke. This *quick return motion* saves time when the tool is not cutting. The crankpin is positioned in or out radially on the bull gear to change the length of stroke. Adjustment is made by turning a shaft that extends outside of the column.

Hydraulic drive. A diagram of a typical hydraulic circuit for shapers and planers is shown in Fig. 9-8. A constant speed motor (2) drives a variable delivery radial piston-type pump (1). The pump delivers oil from the reservoir (3) to a control valve (4) that directs the oil to one side or the other of the cylinder (5). The rate at which the pump delivers oil can be infinitely varied over a wide range. Pressure is available at any speed up to a maximum relief valve setting.

On the cutting stroke, the oil is admitted on the rod side of the piston (7). For the return stroke, oil enters the other end of the cylinder. The oil exhausted from the rod side is carried directly to the opposite side of the cylinder together with that delivered by the pump. That provides a large volume of oil to give the return stroke a high speed.

Comparison of drives. Hydraulic drives offer certain advantages

for shapers as well as other machine tools. The ram or table speed is constant over most of the stroke and does not vary as much as Fig. 9-7 indicates for a crank-operated shaper. Reversal is quick and free from shock because the drive mechanism need not have a high inertia and is cushioned by the liquid. The full capacity of the tool is utilized through most of every stroke. With less time spent in reversal and a high uniform rate of return, more strokes per minute are obtainable from a hydraulic shaper at a specified cutting speed. A comparison for a 40 sfpm cutting speed and a 10 in. stroke showed a hydraulic shaper making 31 strokes per minute and a mechanical shaper 18.7 strokes per minute. When its tool becomes overloaded, a hydraulic shaper has a valve that opens automatically and prevents breaking the tool. Hydraulic speeds are easy to control and adjust as desired.

Mechanical drives are simple and therefore are lower in first cost and easy to service. Many mechanics do not understand hydraulic drives, and specialists may have to be called in to repair them. The length of travel of a shaper tool may vary if the system is hydraulic. Variations in stock cause changes in tool resistance. The speed increases slightly against a lower resistance, and the tool tends to overrun. A mechanically driven tool can go only to the limit set by the mechanism, which is advantageous in cutting to a shoulder.

Available cutting speeds vary with different types and makes of shapers and also with sizes of machines. A 12 in. hydraulic shaper has an infinite number of speeds up to 400 strokes per minute between 5 and 150 sfpm. A 12 in. crank-driven shaper has 8 speeds from 20 to 210 strokes per minute. In general, the larger the size of a shaper, the narrower and lower its speed range.

Ram and tool head. The stroke of a shaper ram can be adjusted for position as well as length of cut. This is done on the crank-driven shaper of Fig. 9-6 through the screw and nut in the ram. The screw is fixed at both ends but can be turned by the short shaft protruding above the ram. The crank and nut move within definite limits for any specific length of stroke. The position of the ram stroke is adjusted by turning the screw to set the ram forward or backward with respect to the nut. After the adjustment has been made, the clamp over the nut is tightened to lock the nut and ram together. By this adjustment, the tool travel is set to match the work surface.

Dogs on the ram of a hydraulic shaper are set to trip a pilot valve

that actuates the control valve to reverse the ram. The settings of the dogs determine the position as well as the length of stroke.

The tool head on the front of the ram of a horizontal shaper has several parts as shown in Figs. 9-1, 9-6, and 9-9. The head may be swiveled to and clamped at any angle in a vertical plane on the front of the ram and is set by means of a scale graduated in degrees. The tool slide can be fed or adjusted at the angle to which it is swiveled. Movement is derived from a screw turned by a crank on top, and settings are obtained from a graduated dial on the screw. The available movement is limited because the tool slide is short. The clapper box on the front of the tool slide carries the toolpost. The clapper box is pivoted and swings to allow the tool to lift on the return stroke. This eases the tool pressure and prevents marring the work surface.

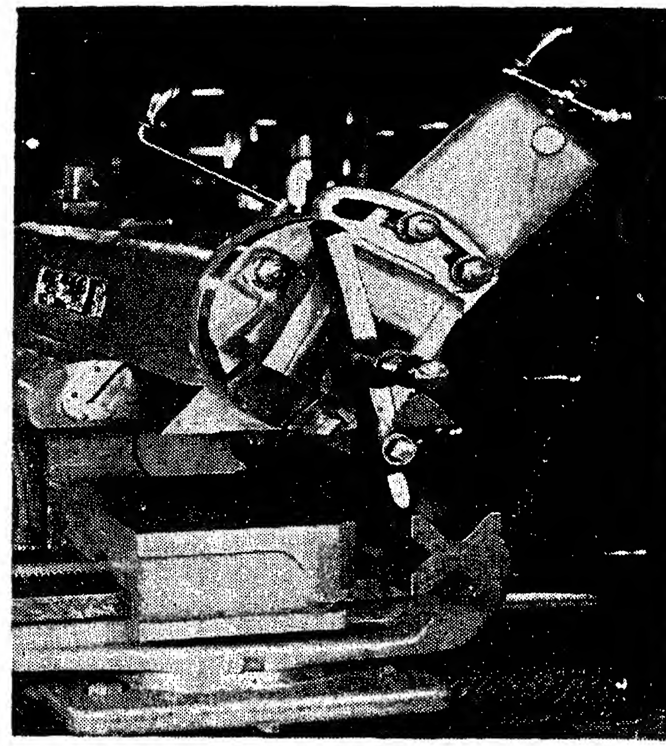


Fig. 9-9. A view of the toolhead and clapper box swivelled on a shaper to machine an inclined surface. (Courtesy The Cincinnati Shaper Co.)

To avoid chatter and provide good backing for the tool, the tool slide should be kept up and the tool allowed to overhang as little as possible. If at all possible, the bottom of the tool slide should not be positioned below the bottom of the ram. The work should be positioned by raising the table to permit taking these precautions.

A tool slide in a vertical position may be fed to machine a vertical surface. The tool slide is adjusted to set a tool to the desired depth of cut when a horizontal surface is cut. If the tool head is placed at an angle, the tool can be fed to shape a surface at the same angle, as depicted in Fig. 9-9. With the tool head at an angle, care must be taken to prevent the ram going too far back and wrecking the head against the ram ways. When a vertical or inclined surface is shaped, the clapper box is swiveled to enable the tool to swing away from the

work properly on the return stroke. Some shapers are equipped with mechanical feed to the tool slide.

Table and cross rail. A cross rail is mounted on ways on the the front of the column of horizontal shapers. The rail is raised or lowered by an elevating screw or screws turned by a crank applied to a shaft at one or both ends of the rail. Usually the cross rail is clamped to the column after being adjusted to set the work to a desired height. Some shapers have a power drive to the elevating screw to raise or lower the rail rapidly.

A saddle slides on ways on the front of the rail and serves to feed the work. The table of a standard shaper is fastened to the saddle, as in Fig. 9-1. The table of a universal shaper can be swiveled through 360° on the saddle. In addition, the universal table has a top that can be tilted 15° either side of center, as shown in Fig. 9-2.

A shaper table has T slots on its top and side for clamping workpieces, fixtures, and vises. A piece is clamped to the top of a table in Fig. 9-10. A long workpiece like a shaft may be clamped to the side of the table to have an end machined. Sometimes fixtures are used, but a swivel vise on top of the table is suitable for most work.

The table of a heavy duty shaper has a support for the end of the table as shown in Figs. 9-1 and 9-2. The support is adjustable for table height and still allows the table to be fed crosswise.

Table feed. The depth of cut on a horizontal shaper is the distance the tool is set below the rough surface. The feed is the distance in inches the table moves across the front of the machine at each stroke or that the tool moves at right angles to its stroke if it is fed by the tool slide. Table feed takes place during the return of each stroke before the tool starts to cut, and the table remains still during the cut. A crank is placed on the end of the cross-screw shaft to feed or position the table by hand. A micrometer dial on the shaft is gradu-

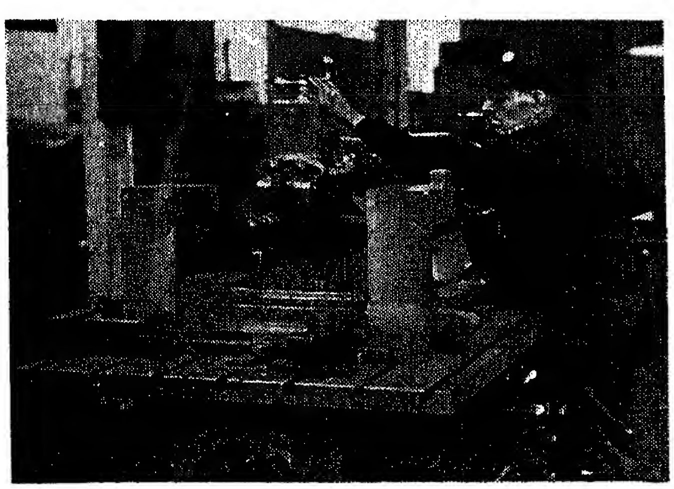


Fig. 9-10. Machining bases for woodworking machines with an offset tool and supplementary table top. The work is clamped to the top of the table. (Courtesy The Cincinnati Shaper Co.)

ated for 0.001 in. increments. Some shapers are equipped with a power rapid traverse drive to move the table rapidly from one position to another.

Power feed is taken off the bull gear shaft of a crank-driven shaper. The cross screw is made to turn a certain amount at each stroke depending on the feed selected. Several kinds of mechanisms are used for that purpose.

On a hydraulic shaper oil under pressure is introduced at either end of the ram stroke into a feed cylinder labeled (10) in Fig. 9-8. This pushes a piston that rotates the driving member of a roller clutch. The driven member of the clutch turns the cross screw. The amount of feed is selected by limiting the length of stroke of the piston by handwheel (12).

Feed ranges vary with sizes and makes of shapers. On most shapers feeds are available from 0.010 to 0.100 in. per stroke in either direction. On some shapers, feeds go higher or lower. Usually 10 to 15 feed rates are provided with mechanical drives. Feeds are infinitely variable within limits with a hydraulic drive.

Shaper Tools and Attachments

Cutting tools. The single point tools used on horizontal shapers are much like lathe tools, except for relief angles. The side relief of a turning tool must be large enough to allow for the helix angle of the cut, but there is no feed on the shaper while the tool is cutting. Also, a lathe tool may be adjusted above or below center on a rocker rest to change the effect of the angles somewhat, but a shaper tool cannot. A shaper tool can have a smaller relief angle than normally ground on a lathe tool. The small relief angle leaves a maximum amount of material to back up the cutting edge. Relief angles of about 4° are recommended for cast iron and steel. For soft materials like aluminum, an acute cutting edge is desirable and the relief angle may be increased to equal the angle of a lathe tool.

Tool bits and holders like those of Figs. 4-11 and 4-12 are commonly used for external shaping. Specific applications are shown in Figs. 4-11 and 9-9. Large tools with integral shanks may be held directly in the toolpost as in Figs. 9-12 and 9-15. A broad nose finishing tool is sometimes used on cast iron. This tool has a straight

front cutting edge ¼ in. or more long set parallel to the surface cut. It takes light cuts not over 1/64 in. deep, and the work is fed at each stroke an amount almost equal to the width of the tool. For internal cutting as is done on the keyway of Fig. 9-11 when a slotter is not

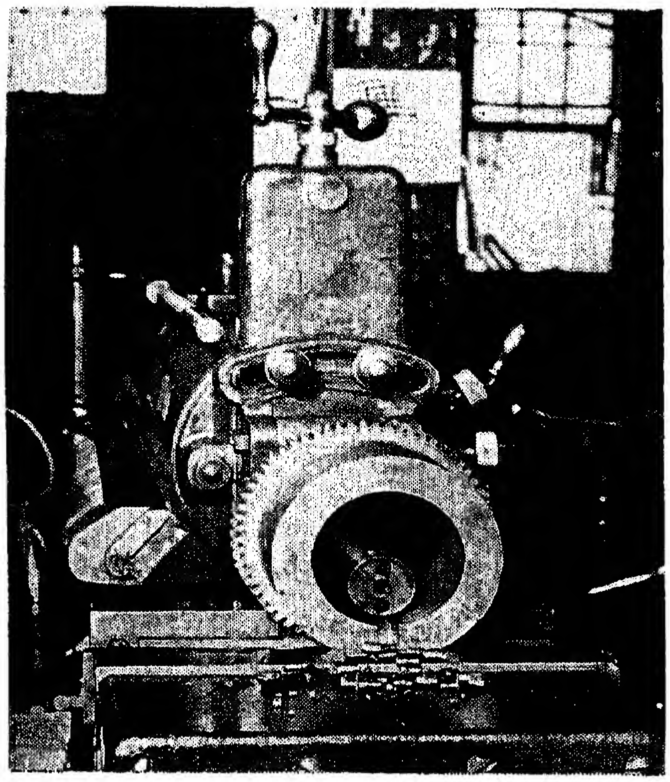


Fig. 9-11. Cutting an internal keyway on a horizontal shaper. (Courtesy The Cincinnati Shaper Co.)

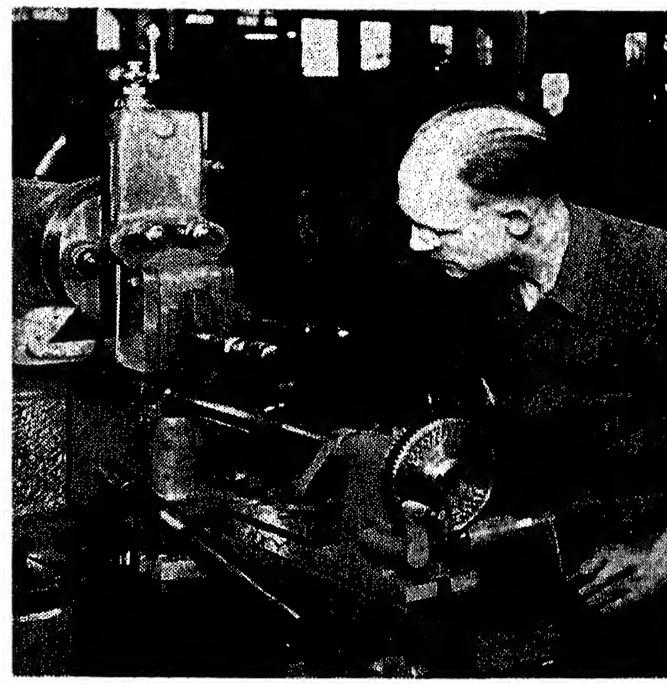


Fig. 9-12. A splined shaft being machined on a shaper with index centers mounted on the table. (Courtesy The Cincinnati Shaper Co.)

available, the tool bit is held by an extension holder or bar that projects into the hole.

A typical solid tool for a vertical shaper or slotter is sketched in Fig. 4-6. The tool moves axially and has rake ground on its end. Tool bits are held in bars or holders, as in Fig. 9-5.

High speed steel is the most common cutting tool material for shaping but shockproof grade carbides are in wide use for shaping alloy cast iron glass molds, alloy steel die blocks, shear blades, wood bits, and other parts of hard materials.

Attachments. *Index centers* provide means for spacing cuts accurately around a workpiece. A splined shaft is being shaped in Fig. 9-12. The workpiece is turned by means of the worm and wheel and is indexed by the pin that registers in holes in the face of the worm wheel.

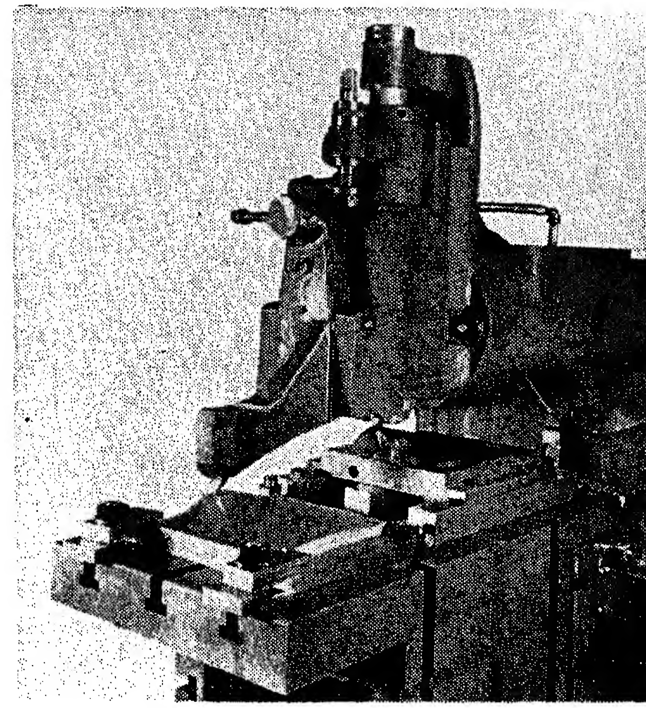


Fig. 9-13. A profiling arrangement on a shaper. (Courtesy Rockford Machine Tool Co.)

Profiling attachments are added to shapers to make them capable of reproducing surfaces of many shapes. A finger or follower is arranged to follow a master surface in unison with the tool passing over the work surface. As the finger rises and falls in tracing the master surface, the tool is raised and lowered hydraulically. Figure 9-13 illustrates a shaper equipped with a special contour head controlled by a hydraulic pressure follower. Often the tracer is mounted on a side head in a manner similar to that shown on the planer in Fig. 10-10.

A *tool lifter* is a device that automatically raises the clapper box and clears the tool fully on the return stroke. It is particularly desirable for use with carbide tools.

A *circular feeding head* is placed on the front of the ram and can be rotated by hand or automatically for shaping concave work or internal circular surfaces with interruptions.

Shaper Operations

Setup. The tops of a standard shaper table and vise are parallel to the movement of the ram. The face of the fixed jaw of the vise is square with the table top. The normal alignment of most shapers is close enough for locating ordinary work. When precise work must be done, the accuracy of the machine may have to be tested and corrected. To check the table or top of the vise body, an indicator on a rod clamped in the toolpost is run along a parallel on the work locating surface. To check the squareness of the vise jaw, the indicator may be run along the edge of a square having its head clamped against the jaw. If the indicator shows a variation, paper shims are placed between the table and vise to correct the misalignment.

An indicator may be used to get more accurate settings than can be obtained from the angle scales on the vise or tool head. To set the tool head in a vertical position, an indicator attached to the tool-post is run along the edge of a square with its head on the table. To set the tool head at an angle, the indicator can be run along a bevel protractor on the table.

A workpiece is held in a vise on a shaper so that the cutting force is directed against a jaw if possible. That provides backing for the piece and decreases the possibility of slipping. The tool should preferably travel lengthwise along a surface. In that way fewer strokes are needed to cover the surface, and less time is lost in return strokes. The surface being shaped should extend above the jaws to avoid running the tool into the jaws. Workpieces often are rested on parallels. Copper, brass, or lead strips called *false jaws* are inserted at the sides of a rough piece as indicated in Fig. 9-14 A to protect the vise jaws.

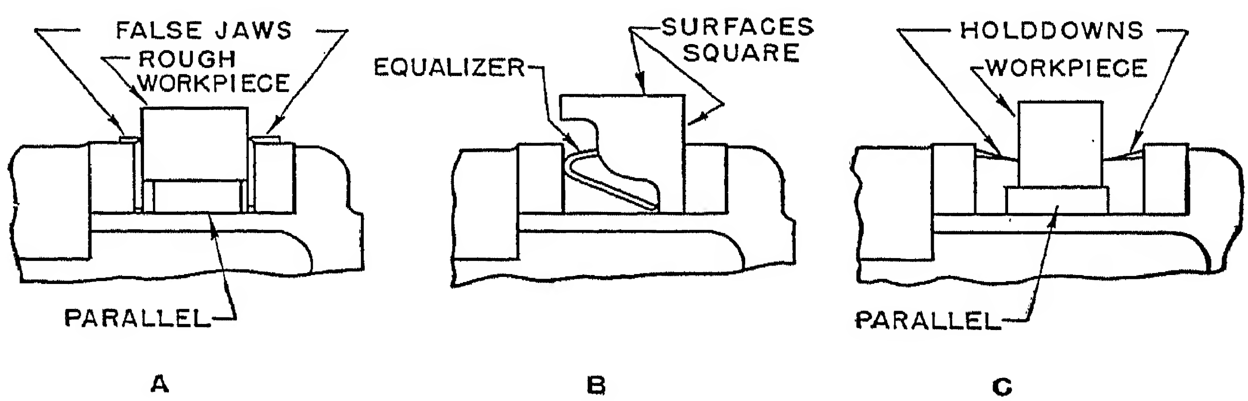


Fig. 9-14. Ways of holding work in a vise.

When a surface is to be shaped square with another, the locating surface is held against the fixed jaw of the vise. An equalizer of bent flat stock between the movable jaw and workpiece stablizes the location as indicated in Fig. 9-14 B. Sometimes a rod will do as an equalizer. If a vise is not used, the locating surface may be squared by clamping it to the face of an angle plate on the table. It the top of a piece is to be machined parallel with the bottom, the piece is held down firmly on the vise body or parallel by *hold downs* as in Fig. 9-14 C. These are thin tapered strips tilted at an angle.

Very small tolerances can be held on a shaper if sufficient skill and care are exercised by the operator. These may be of the order of 0.001 in. or smaller on dimensions and 0.001 in. in 6 in. for square-

ness and parallelism. However, more usual and economical tolerances are 0.003 in. to 0.005 in. on dimensions and 0.002 to 0.003 in. in 6 in. for squareness and parallelism.

Large or irregular pieces are often marked by layout methods described in Chapter 3 to locate the surfaces to be shaped and insure that the right amount of stock is removed in the proper places. The scribed lines may be accentuated by light prick punch marks. Lines and surfaces on the workpiece may be leveled on the shaper by means of a surface gage. The tool is adjusted to cut an irregular surface by the operator manipulating the controls, as exemplified in Fig. 9-15. Cast iron is prone to chip off where the tool leaves the cut, making a jagged edge and defacing any line on the end surface. This tendency is reduced by putting a large bevel on the end edge of the workpiece.

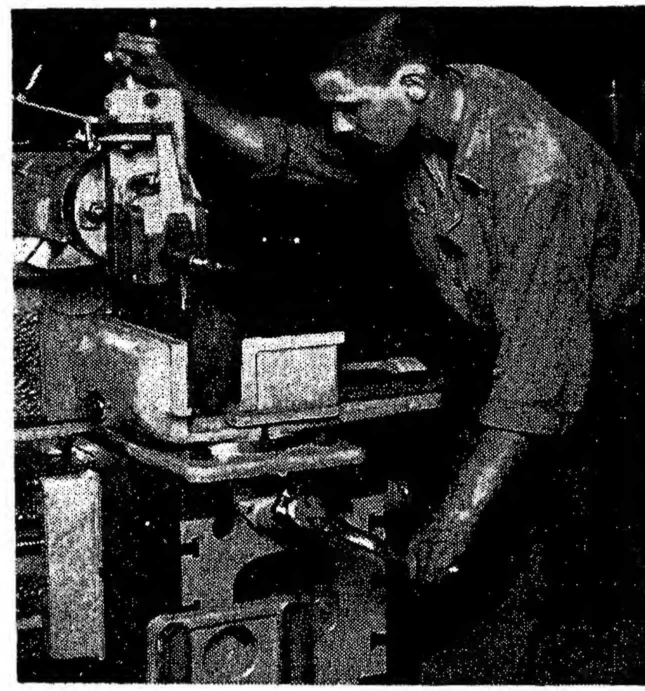


Fig. 9-15. An operator guiding the tool on a shaper by manipulating the vertical and cross feeds. This machine has an auxiliary crossfeed control on the front. (Courtesy The Cincinnati Shaper Co.)

Shaping compared with other operations. Other machine tools are able to cut and remove stock faster than shapers, but shapers are favored for many short-run jobs because of several advantages they offer. They are able to machine almost every kind of surface on small and moderate size parts. A shaper can be put to many uses in a shop and can easily be changed from one job to another. Setup time for many jobs is less on a shaper than on other machines. Practically all work can be done with simple and inexpensive tools without extra attachments. This is exemplified by any irregular surface that would require a special form tool if machined by another method such as milling, but can be shaped with a single point tool. Even if a form tool is used on a shaper, it can be made and sharpened more easily than a multiple tooth milling cutter or broach.

A shaper is convenient for cutting inclined surfaces. On other

machines such as the milling machine, fixtures and attachments often are needed to machine inclined surfaces. Frail workpieces can sometimes be shaped with ordinary setups because the cutting forces are not large, whereas extra supports would be needed on other machine tools.

Estimating shaper time and power. The speed of a shaper is expressed in strokes per minute. The problem is to find the machine speed, N, in strokes per minute that will give the tool the correct cut speed, C, in sfpm for the desired length of stroke, *l*, in inches. The cut speed on each stroke increases from zero to a maximum and then returns to zero. The average cut speed is a little less than the highest cut speed. The cut speed of a single point tool on a shaper should be about the same as on a lathe under comparable conditions.

The cutting stroke occupies only part of each full stroke. The proportion varies with different shapers, but the ratio of 1.6 to 1 in Fig. 9-7 is about average. For that case, the tool is moving at cut speed during 61 per cent of a full stroke. If it were to continue at the same average rate for a full stroke, the tool would cover a distance of $l/0.61$ in. Thus

$$N \times \frac{l}{0.61} = 12C \qquad \text{and} \qquad N = 7.3\,\frac{C}{l}$$

Long strokes and high speeds together impose a strain on a mechanical shaper and should be avoided. Most shapers are not capable of delivering cutting speeds as high as those used in turning aluminum.

A machine steel surface 10 in. long is to be shaped with a HSS tool at 80 sfpm. The shaper should be set as near as possible for $7.3 \times (80/10) = 58.4$ strokes per minute.

The feed on a shaper should be as heavy as the machine will stand and as will not give too rough a surface. Table V shows average feeds for cast iron and machine steel. Higher or lower feeds may be selected to suit particular situations. The feed may be reduced to ½ or ⅓ for irregular surfaces, grooves, and cutting off. The finer the feed, the better the finish. Softer materials allow higher feeds, and hard materials require lower feeds. A rigid setup permits a heavier cut than a weak one. A feed as high as ¼ in. per

stroke may be desirable for a broad nose tool, and one as fine as 0.005 in. per stroke for a sharp point tool.

Table V

Type of cut	Roughing				Finishing	
Depth or manner of cut	1/16	1/8	1/4	Tool head feed	1/32	Tool head feed
Feed in./stroke	0.062	0.037	0.025	0.020	0.025	0.010

The feed rate in inches per minute is the product of the feed in inches per stroke times the number of strokes per minute. The distance of feed is the width of the surface cut plus the amount of overtravel. The time required for a cut is the quotient of the distance the work or tool is fed divided by the feed rate in inches per minute.

A machine steel surface 10 in. long and 5 in. wide is to be shaped at 60 strokes per minute. The depth of cut is $\frac{1}{8}$ in. and the feed 0.037 in. per stroke. The feed rate is $0.037 \times 60 = 2.22$ in. per minute. If $\frac{1}{8}$ in. overtravel is assumed, the distance the work is fed is $5\frac{1}{8}$ in., and the machine time is $5.125 \div 2.22 = 2.31$ minutes.

The maximum rate of metal removal in shaping is found by multiplying the feed in inches per stroke by the depth of cut by the surface speed in sfpm by 12. In the foregoing example, the rate of metal removal is $0.037 \times \frac{1}{8} \times 82.3 \times 12 = 4.56$ cu in. per min. For a unit power consumption of 1 hp per cu in. per min, 4.56 hp is expended at the tool point. For a machine efficiency of 75 per cent, 6.1 hp is required at the motor. Typical motor specifications are 5 hp for 16 and 20 in., $7\frac{1}{2}$ hp for 24 in., and 10 hp for 36 in. shapers.

Questions

1. How is the size of a shaper designated?
2. How does a horizontal shaper differ from a vertical shaper or slotter?
3. How does a standard shaper differ from a universal shaper?
4. What is the difference between a geared shaper and a gear shaper?
5. What is a draw cut shaper, and what advantages does it offer?
6. What are the advantages of a vertical shaper or slotter?
7. Describe the mechanism of a crank-operated shaper.

8. Describe the driving system of a hydraulic shaper.
9. What are the relative advantages and disadvantages of crank-operated and hydraulic shapers?
10. What cutting tools are used on shapers?
11. How may the units of a shaper be aligned and adjusted accurately?
12. How is a workpiece held in a vise to shape one surface square with another? To shape one surface parallel with another?
13. Tell how an irregular surface may be shaped.
14. Upon the basis of what considerations should a shaper be selected for a job?

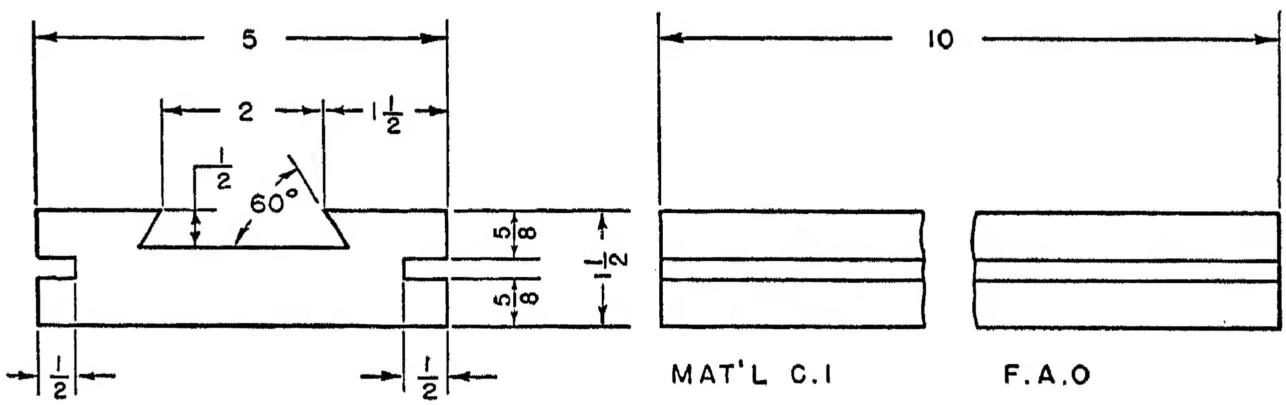


Fig. 9-16. A slide block.

Problem

Write an operation instruction sheet to make one piece specified in Fig. 9–16. The rough casting has ⅛ in. stock on all surfaces. Normal operation tolerances are satisfactory.

References

Brown, H. W., "How to Estimate Cost of Shaper Jobs," *Machinery*, Aug. 1949, Vol. 55, No. 12, p. 152.

Nordhoff, W. A., *Machine Shop Estimating*. New York: McGraw-Hill Book Co., Inc., 1947.

Manufacturers' Catalogs:
- Cincinnati Shaper Co., Cincinnati, Ohio.
- Morton Manufacturing Co., Muskegon Heights, Mich.
- Pratt and Whitney Division, Niles-Bement-Pond Co., W. Hartford, Conn.
- Rockford Machine Tool Co., Rockford, Ill.
- Smith and Mills Shapers, Inc., Division Hamilton-Thomas Corp., Hamilton, Ohio.

Chapter 10

PLANERS

THE PLANER MACHINES SURFACES BY SUCCESSIVE cuts like the shaper, but its construction and action enable it to do heavy work on large pieces. The work is carried on a massive, fully supported, and horizontal table capable of sustaining heavy loads. The table reciprocates the work past one or more tools that are fed for each

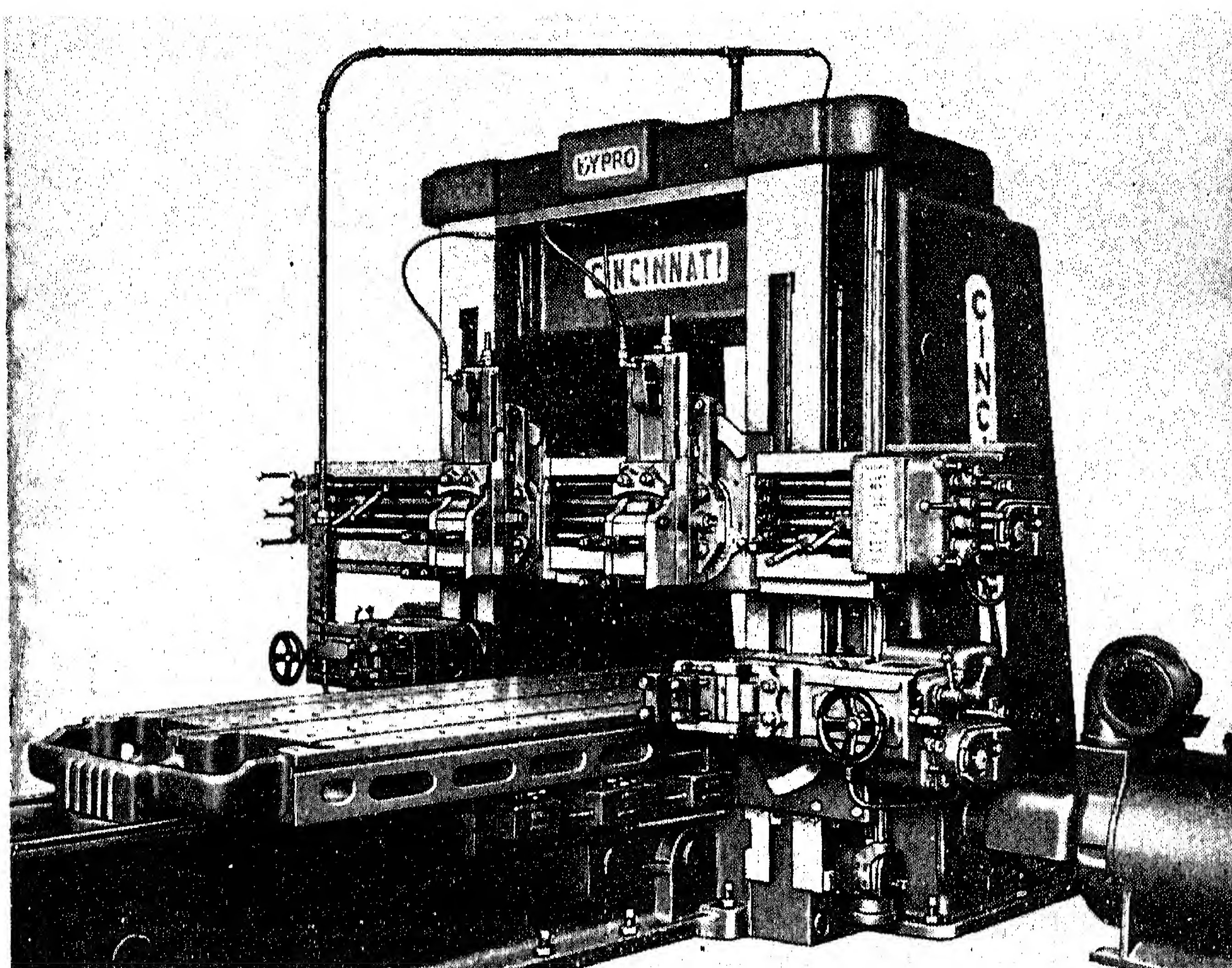


Fig. 10-1. A 48 in. by 48 in. by 16 ft double housing planer with twin helical gear drive. (Courtesy The Cincinnati Planer Co. Div., Giddings and Lewis Machine Tool Co.)

stroke. The tools are held by heads mounted on slides on vertical columns or a horizontal cross rail.

Types and Sizes of Planers

Standard or double housing planer. The standard or double housing planer is capable of heavy service. The cross rail over the

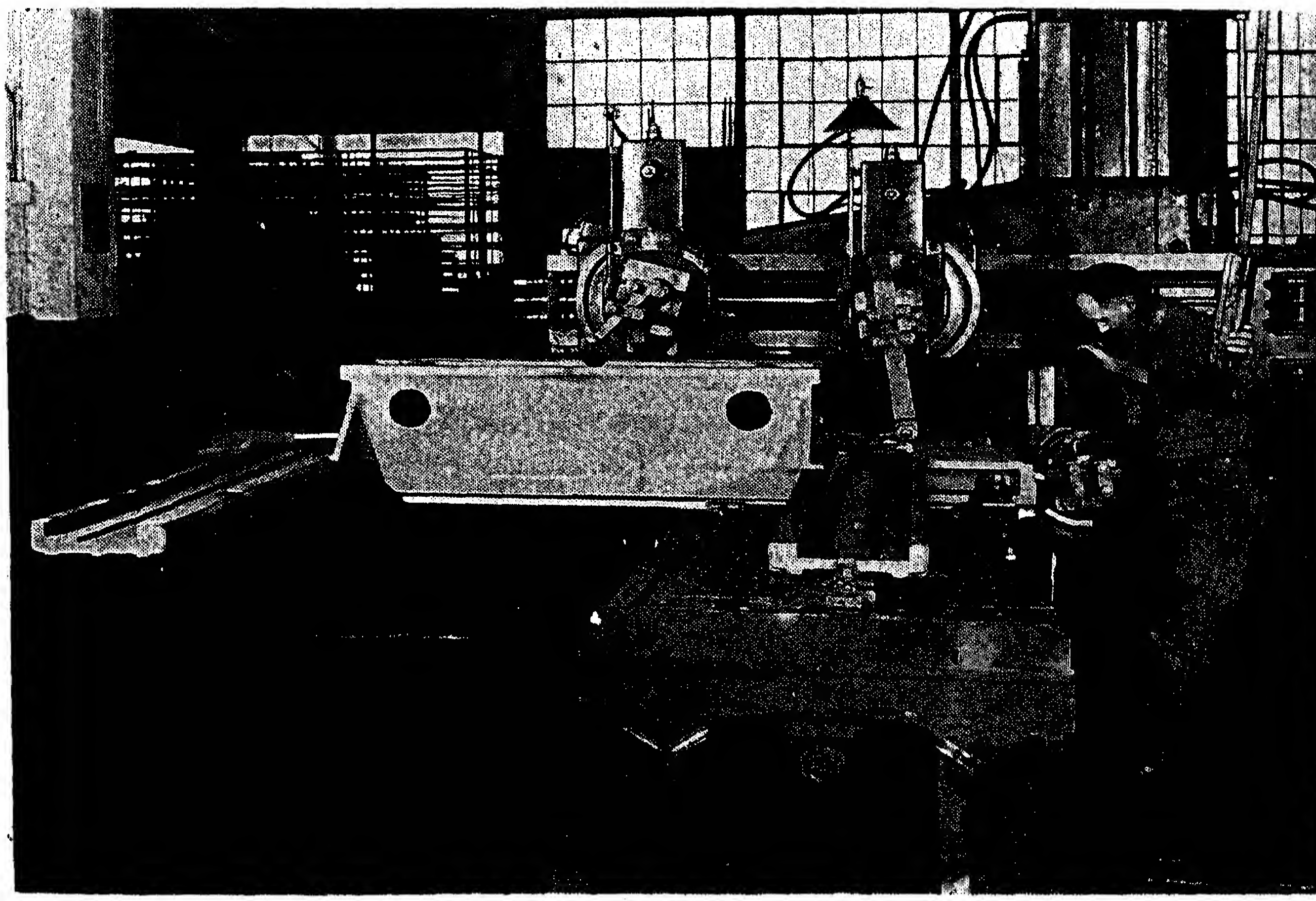


Fig. 10-2. An openside hydraulic planer with a workpiece on the table extending beyond the rail. (Courtesy Rockford Machine Tool Co.)

bed is carried on two vertical housings or columns tied together on top as shown in Fig. 10-1. This type of machine may have one, two, or three tool heads, but is mostly equipped with four.

Openside planer. An openside planer has a housing only on one side of the bed. The other side is open so that extra wide work can hang over the side of the table, as shown in Fig. 10-2. With the exception of one housing and tool head, an openside planer generally has the same components as a standard planer. Very wide work may be supported also on an auxiliary rolling table alongside

and parallel to the table of the machine. Small sizes of openside planers are also called *shaper planers* and generally have two tool heads, one on the housing and one on the cross rail.

Fig. 10-3. An hydraulic planer type grinder working on a railroad crossing. (Courtesy Rockford Machine Tool Co.)

A *convertible openside planer* has a removable housing that can be fastened to the bed to support the outer end of the rail. The extra housing may have a tool head. That machine offers the convenience of an openside planer for wide work plus two-column strength for work that is not too wide to pass between the columns.

Planer millers and grinders. The *planer-type milling machine* is a cross between the planer and milling machine and is described in Chapter 12.

Workpieces like railroad crossways, frogs, and switches of hard-to-cut material must be ground. An openside planer-type grinder is shown in Fig. 10-3, with a grinding head instead of the conventional tool head on the rail.

Plate planer. A plate planer is a special-purpose machine for squaring and beveling the edges of steel plates for armor, ships, or pressure vessels. A plate planer capable of taking work up to 32 feet wide is illustrated in Fig. 10-4. One end of a plate being machined is held down by the air-operated clamps suspended from the overhead bridge. The remainder of the plate may be supported by a ball table that has a number of steel bal's on uprights. The tool is traversed by the carriage on the front of the machine, is held in a single revolving toolholder so that cutting can take place in either direction, and can be adjusted horizontally and vertically. The machine shown can handle work up to about 8 in. thick.

Pit planer. When workpieces are too heavy or bulky, the tools may be moved more readily than the work. That is the basis for the design of the pit planer of Fig. 10-5. The workpiece is mounted

on a stationary table. The columns carrying the cross rail and tool heads ride on long ways on both sides of the table.

Sizes of planers. The size of a standard planer is designated by the distance between the vertical housings in inches, the height from the top of the table to the rail in its uppermost position in inches, and the maximum length of table travel in feet. The length of travel is often the same as the working length of the table. The dimensions indicate the size of the largest workpiece that can be accommodated. Thus, a 42 in. by 42 in. by 10 ft planer can machine a workpiece with sides of those dimensions.

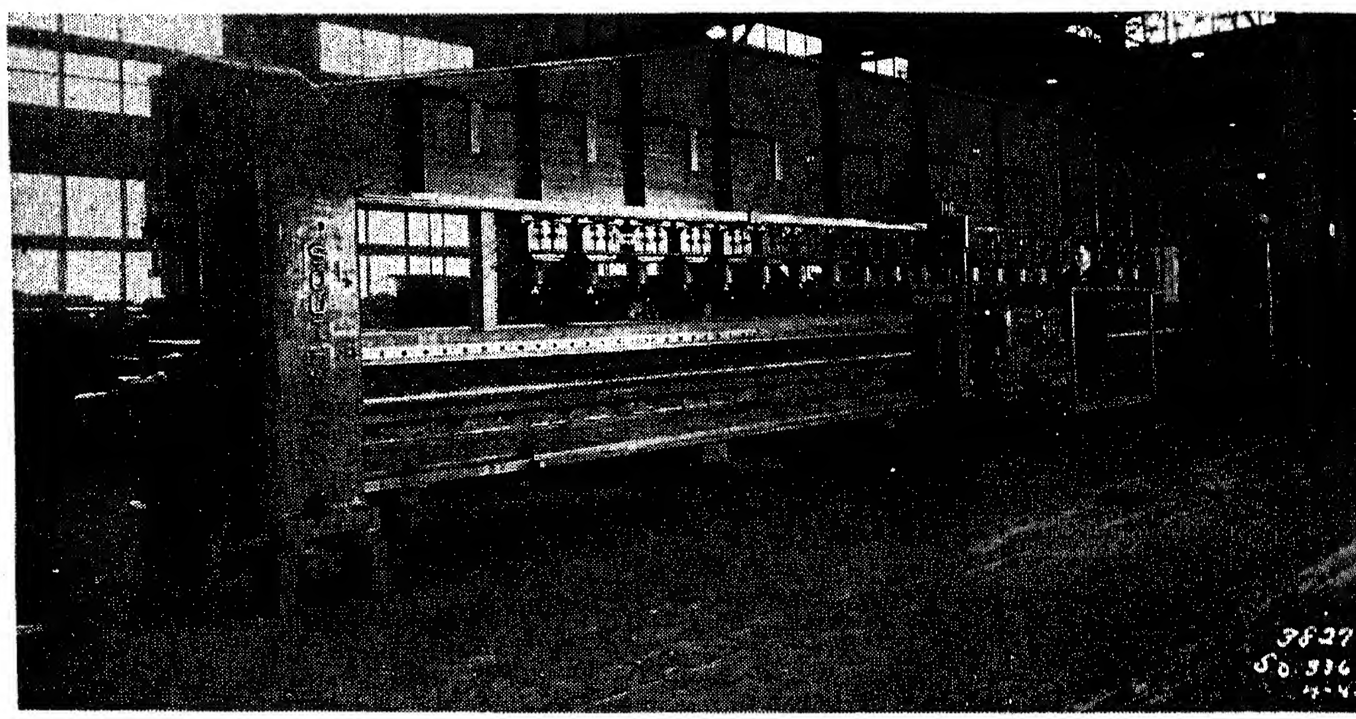


Fig. 10-4. A 32 ft plate planer. (Courtesy Baldwin-Lima-Hamilton Corp.)

Sometimes only the first two dimensions, the width and height, are given. If they are the same, one figure may represent both. Thus, a 42 in. by 42 in. machine may be called a 42 in. planer. Planers with a particular size opening over the table usually are available with various lengths of table travel. For example, 42 in. by 42 in. planers are made with tables from 10 ft to 25 ft in length.

Common sizes of standard planers range from 24 in. to 120 in., although some machines have been made for work as large as 16 ft wide by 16 ft high by 60 ft long.

The height and length of openside planers are specified in the same way as for standard planers. Obviously, the width dimension

cannot be the distance between columns. Instead, a dimension called the planing width determines the sizes. It is the extreme distance from the table side of the column to the tool in the outer tool head in a vertical position. The figure given for the width of the machine is approximately 12 in. less than the planing width. Thus, a 48 in. by 48 in. by 12 ft openside planer has a planing width of about 60 in., a maximum clearance of 48 in. between table and

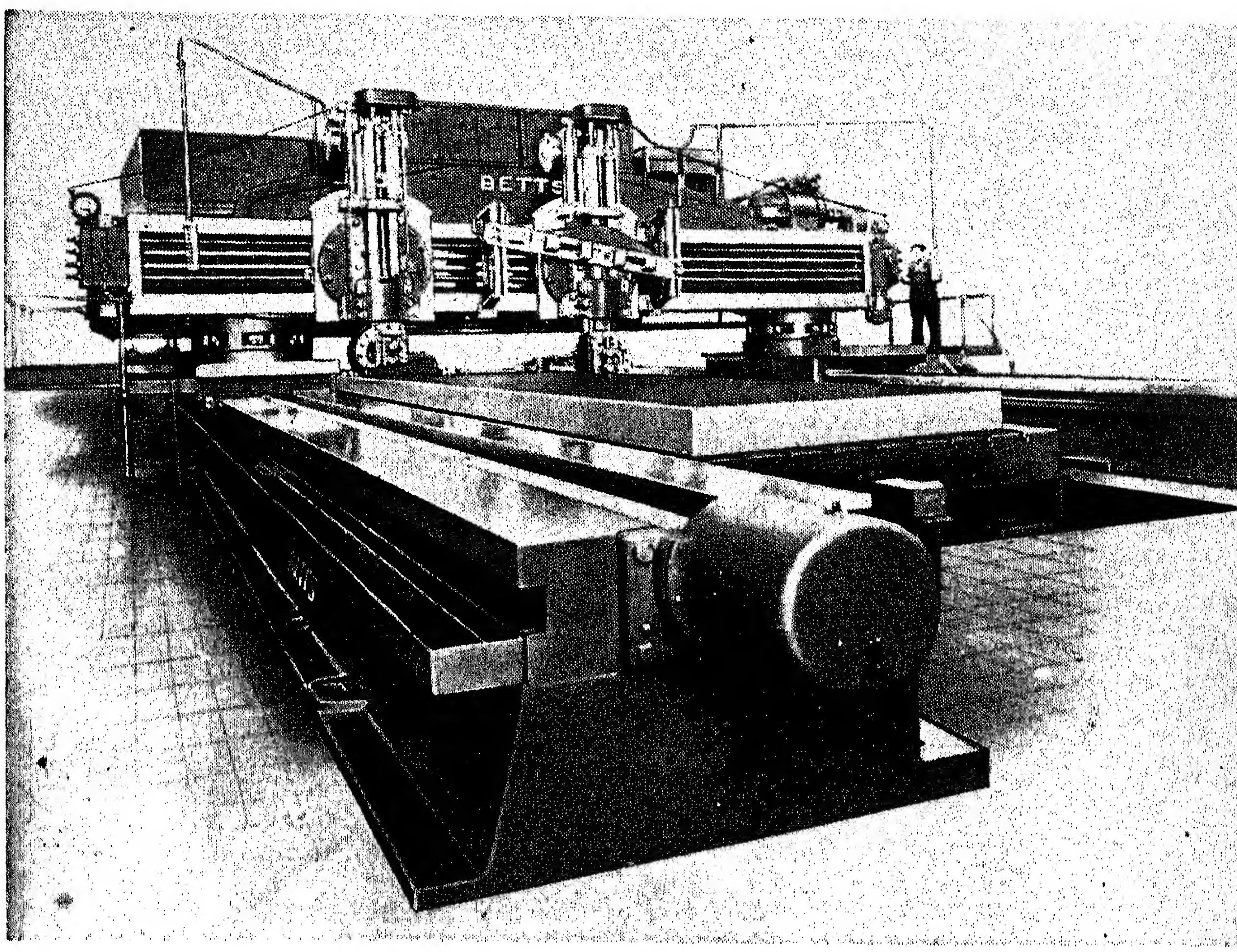


Fig. 10-5. A pit planer. (Courtesy Consolidated Machine Tool Corp.)

rail, and a table travel of 12 feet. Standard openside planers are made in sizes from 24 in. to 120 in.

Elements of planers and Their Functions

Table and bed. The table of a conventional planer is a heavily ribbed boxlike casting that slides on accurate V ways on a massive and rigid bed. Most ways are machined and scraped cast iron surfaces. They are force lubricated with an ample supply of oil. On

some machines the table drive is shut off if the oil supply to the ways fails. The film of oil builds up when the table is moving and "floats" the table. If the table stops, it settles slightly, and the result may be a step in the surface being machined. Thus, the tools must be checked before a job is started to make sure they will last until the surface is finished.

The bed of a modern planer is about twice as long as the table to support fully the table in all positions. Very long beds may be made in sections carefully machined and bolted together. Most planers have two V ways, but some may have several ways, V and flat, especially if the table is wide. Guides along the V ways take any side thrust that tends to lift the table out of the V ways.

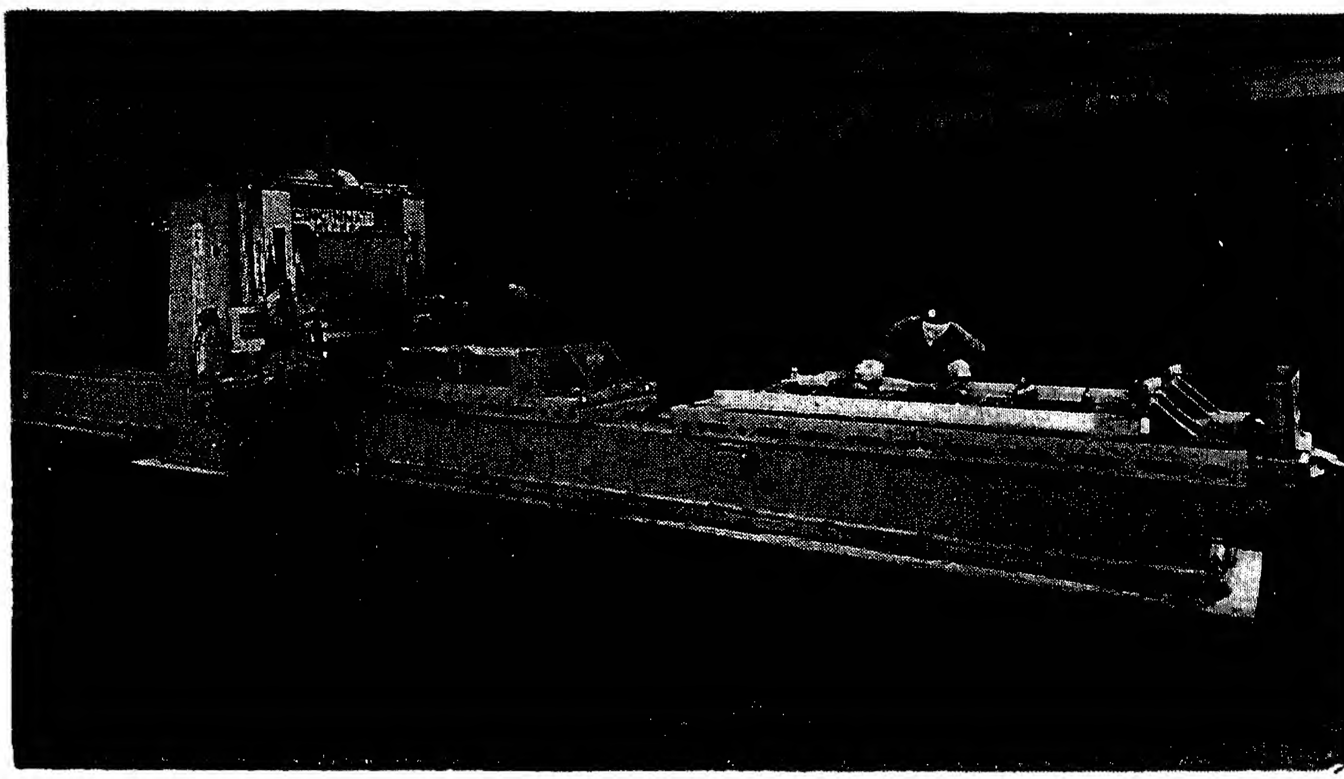


Fig. 10-6. A planer with a divided or latching table. One section is carrying work being planed, the other is being set up. (Courtesy The Cincinnati Planer Co. Div., Giddings and Lewis Machine Tool Co.)

Planer ways are made straight and parallel to close tolerances, but if the bed is placed on an uneven foundation, it sags, and the ways become distorted and untrue. Leveling jacks or screws are commonly provided on the bottom of the bed. Leveling is done with a precision spirit level applied to finished surfaces along and across the ways. Sagging and distortion are emphasized in large

machines, such as planers, if they are not level, but all machine tools need to be well supported and level if they are to perform properly.

The top of a planer table is finished to provide an accurate working surface, normally the same length as the stroke. In addition, a chip trough at each end may be used to support overhanging workpieces. T slots run the entire length of the table to take hold down bolts. Some tables have renewable steel inserts in the T slots. Between the T slots are rows of reamed holes for stop pins and clamps.

Most planer tables are made in one piece, but some have two sections. These are found on *divided* or *latching table planers.* The sections may be coupled together for long work or separated so that the work on one table may be set up while that on the other is machined, as depicted in Fig. 10-6.

A heavily loaded planer table might very well run off the end of the bed and cause considerable damage if for some reason it were not reversed as intended at the end of a stroke. Many planers have safety devices to take care of just such an emergency. Among these devices are hydraulic bumpers. Another form has cutting tools bolted to the underside of the table. If the table overruns, the tools bite into replaceable stop blocks fastened to the bed and dissipate the kinetic energy of the moving table.

Table drive. Some planners have hydraulic drives, others, mechanical drives. Modern planers with mechanical drives are powered by variable speed, reversing, d-c motors. A motor generator set powered from an a-c supply furnishes adjustable voltage d-c to the motor of the machine. The main drive motor would ordinarily have a speed range characteristic in the ratio of 3 to 1 or 6 to 1. With the adjustable voltage feature, the speed range has a ratio of 30 to 1. Thus the realizable speed range of the motor is normally from 40 to 1200 rpm, with practically a constant horsepower delivered over the range of 500 to 1200 rpm. The forward and reverse speeds are separately controlled. Even with the larger motors, reversal is almost instantaneous. Motors range in rated sizes from 15 to 25 hp for 30 in. planers to 50 to 75 hp for 72 in. planers.

The power is carried from the motor to the table through a set of reduction gears housed in the bed. The last gear in the series is a bull gear that meshes with a rack fastened to the underside of the

table. Herringbone or twin helical gears and racks are favored to eliminate side thrust.

The gear ratio of a planer is defined as the number of turns the motor shaft makes to drive the table one foot. Gear ratios vary from 4 to 12½, and each planer is made with a specific ratio. The higher ratios are put on heavy-duty machines. If the gear ratio is 4 and the motor speed range 40 to 1200 rpm, the table speed range is 10 to 300 sfpm. With a gear ratio of 8 and the same motor, another planer has speeds from 5 to 150 sfpm but twice as much table pull at any specified speed.

On a hydraulic planer, a constant speed a-c motor drives the hydraulic pump. Hydraulic planers are powered with 40 hp and 60 hp units. A typical hydraulic circuit is described in Fig. 9-8 and Chapter 9. The maximum table speed varies with the type of service for which the machine is designed and may be from 50 to 300 sfpm on different machines. Faster return speeds independent of the cutting speeds are available. The same advantages and disadvantages of hydraulic drives cited for shapers apply to planers.

Adjustable dogs bolted to the side of a planer table and located by a slot are set to reverse the table at the end of the desired stroke. The dogs trip a switch in the electrical circuit on a mechanical planer or actuate a control valve on a hydraulic planer.

Housings and rail. The housings, also called columns and uprights, are the heavy structural members fastened to the sides of a planer bed and carrying the horizontal cross rail. The vertical ways on the housings may carry side tool heads in addition to the rail as shown in Fig. 10-1. Another type of construction, depicted in Fig. 10-2 and found on double housing as well as openside planers, provides an apron attached to the rail and bearing over most of the housing ways. Each apron has its own set of ways for a side tool head. Thus, with this construction, all tool heads are actually carried on the rail, and the rail alone slides on the housing or housings. The vertical ways on the rail apron can be swiveled to make them square with the top of the table.

The housings contain elevating screws, feed and rapid traverse rods, and other parts of the drive mechanism for the cross rail and tool heads. The rail is counterbalanced and raised or lowered by a screw in each housing by power or by hand. The screws on a double housing planer are synchronized so that the rail is always

kept level. Manual, electric, or hydraulic rail clamps act on the ways to hold the rail securely when it is not being moved. The clamping force is equalized, and interlocks prevent moving the rail when it is partially or fully clamped. Each side head is raised and lowered by a screw turned by power or by hand and can be clamped when desired.

The cross rail carries one or two tool heads, called rail heads. The rail is made long enough to move one of a pair of tool heads out of the way to allow the other to take a full cut across the table. The rail contains the cross screws for moving and feeding the rail heads and feed rods to carry power to the heads. The heads may be moved by power or by hand and can be clamped when desired.

Tool heads. A planer tool head consists of a saddle, swivel plate, slide, and clapper box. These components are seen on the machines of Figs. 10-1 and 10-2. The saddle slides on the ways on which the head is mounted. Tool-head saddles are made right- and left-handed. Two of opposite hands are put on the cross rail so their slides can be run close together. The hand of each side head is such as to permit its slide to approach close to the cross rail. The swivel plate on a tool head can be swiveled and clamped at an angle, and the slide fed by power or by hand at that angle. The swivel plate is usually graduated 60° on each side of normal position. The clapper box can be swiveled up to 20° on either side on the slide and allows the tool to rise from the surface on the return stroke. Many planers have air or hydraulic tool lifters that automatically swing the clapper block out for each return stroke. Tools are commonly clamped to the clapper by straps as shown in Figs. 10-2 and 10-6.

Feeds. Feed on a planer is the distance in inches the tool is moved for each stroke of the work. This may result from moving the saddle or the slide for a horizontal or a vertical cut. Two feed movements may occur at the same time, at the same or different rates, to cut an angle. Angles or inclined surfaces may also be cut by swiveling a tool head and feeding the tool slide. The feeds to the various heads are independent of each other. The saddle of one head may be fed at the same time as the slide of another head.

Feed of a planer tool is intermittent and reversible in direction. The feed mechanism of hydraulic planers is described in Fig. 9-8 and Chapter 9. A common feed range is 0.010 to 0.500 in. per stroke,

but a range from 0 to 1 in. per stroke is available on some machines. Any feed rate within the range can be obtained.

The feeds on mechanical planers are stepped. A typical arrangement provides feed rates from 1/64 to 1 in. per stroke in 64ths. The mechanical planer of Fig. 10-1 has an electric dial feed. Small relays interlocked with the dog-actuated contactors in the master reversing switch cause magnetic clutches for each head to engage at reversal of the planer stroke. A measuring unit mechanism for each head automatically interrupts the current after the selected tool movement. The feed rates are selected by turning the star knobs on the measuring units on the end of the rail and each side head.

In addition to the feeds, the rail and the tool-head slides and saddles can be moved independently at a rapid traverse rate. A separate reversible motor provides the power for rapid traverse.

Controls. The main control of modern planers is the *pendant*, shown suspended from the top of the housings in Fig. 10-1 and in the operator's hand in Fig. 10-2. It contains buttons to start and

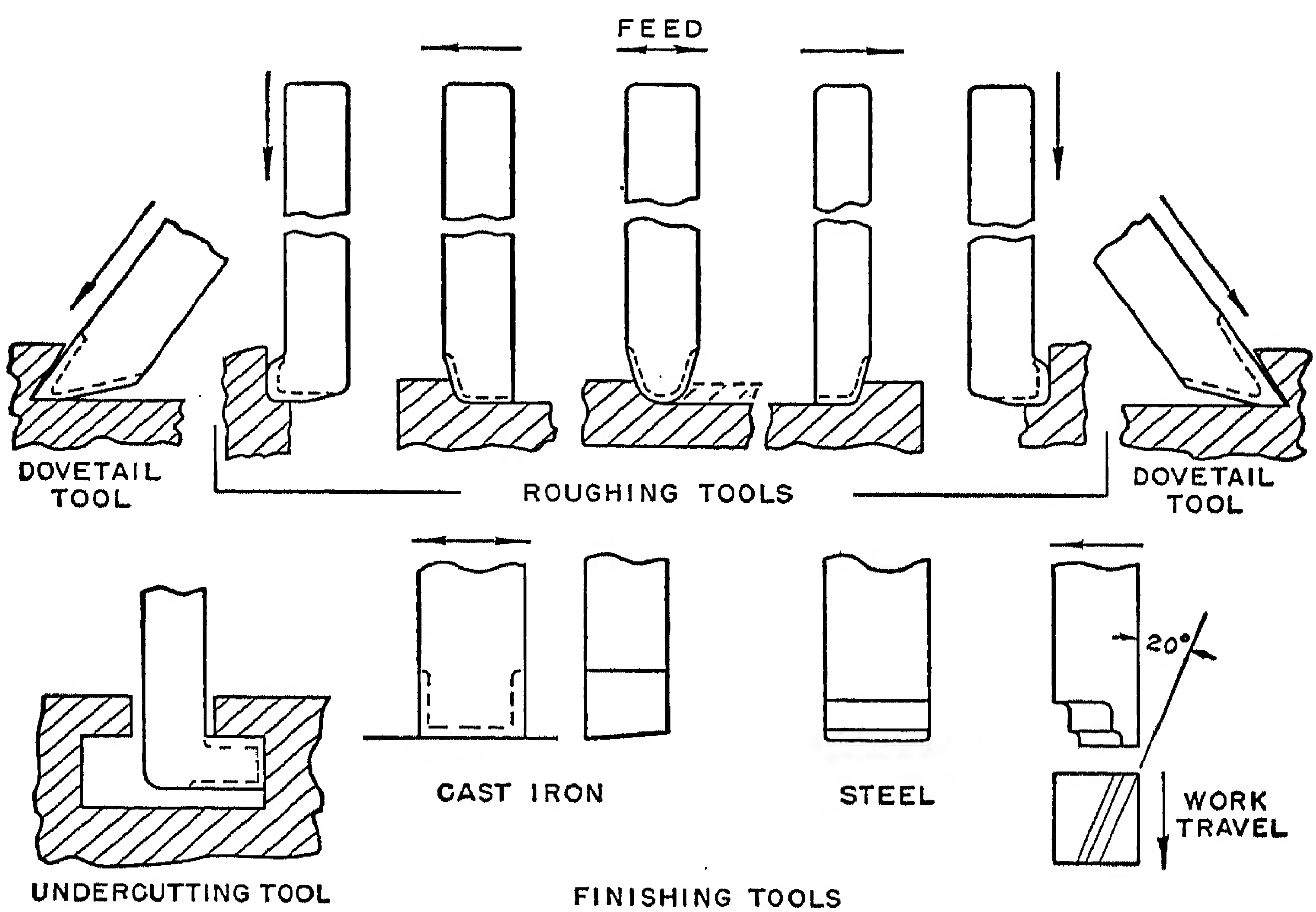


Fig. 10-7. Typical planer tools.

stop the main motor and to move the rail up or down. It may also have an "inch control" for jogging the table in either direction and a "tool-saver control" to slow the table while the tool is passing through a hard spot.

At the operator's position on the right of Figs. 10-1 and 10-2 are controls on the end of the rail for engaging feed or rapid traverse movements of the rail heads, left or right, up or down, separately or together. The rates of feed may be selected also at this position. Many planers are provided with duplicate sets of controls, one set on each end of the rail. Each side head is controlled at its own station. All movements of the tool head may be made by hand and are indicated by micrometer dials.

Planer Tools and Attachments

Cutting tools. Planer tools have the same general shapes and angles as lathe and shaper tools but for the most part are heavier and larger because the planer normally takes deep cuts at heavy feeds. Typical tool shapes are shown in Fig. 10-7. Round nose tools are favored for roughing, but sharp point tools are necessary to get into corners, such as dovetails. For finishing cast iron, bronze, and aluminum, broad nose tools are used with cutting edges from 1 to 1½ in. wide set carefully parallel to the surface cut. For finishing steel, a shearing cut with the broad edge of a tool at an angle of about 20° to the direction of cut is desirable to keep from tearing the metal.

Planer tools like those sketched in Fig. 10-7 may be forged of high speed steel, but for economy the expensive cutting material is welded, cemented, or clamped to a mild steel or cast iron shank. A short piece of high speed steel may be welded on the end of a mild steel shank. A typical commercial planer tool shank to which various shapes of bits are clamped is shown in Fig. 4-12. One end of the tool has a gooseneck form that is advantageous for planing because the tool tends to spring away from rather than to gouge into the surface when the cut is heavy. Clamped tool bits may be removed from the holder and replaced without removing the shank from the machine. Bits may be made of high speed steel, cast alloy, or carbide.

Tips of high speed steel, nonferrous cast alloy, or carbide are cemented to mild steel or cast iron shanks for planer tools. Carbide applications have been quite successful for cast iron. Generally speaking, the high carbon alloys are easier to plane with carbide than are the low carbon or mild steels.

One form of *undercutting tool* is depicted in Fig. 10-7. It is hooked to machine T slots. With such tools, the clapper box must not swing because the tool will be jammed into the overhanging ledge of the T slot on the return stroke. Another design of undercutting tool has a bit pivoted on its holder so that it can swing horizontally in the slot on the return stroke. A *gang tool* has several bits in one holder. Each bit take part of a cut. *Reach tools* are helpful where a long overhang is necessary, as from the right-hand rail head of Fig. 10-2. One form of reach tool has a small tool box with a clapper block to hold the tool bit on the end of the heavy

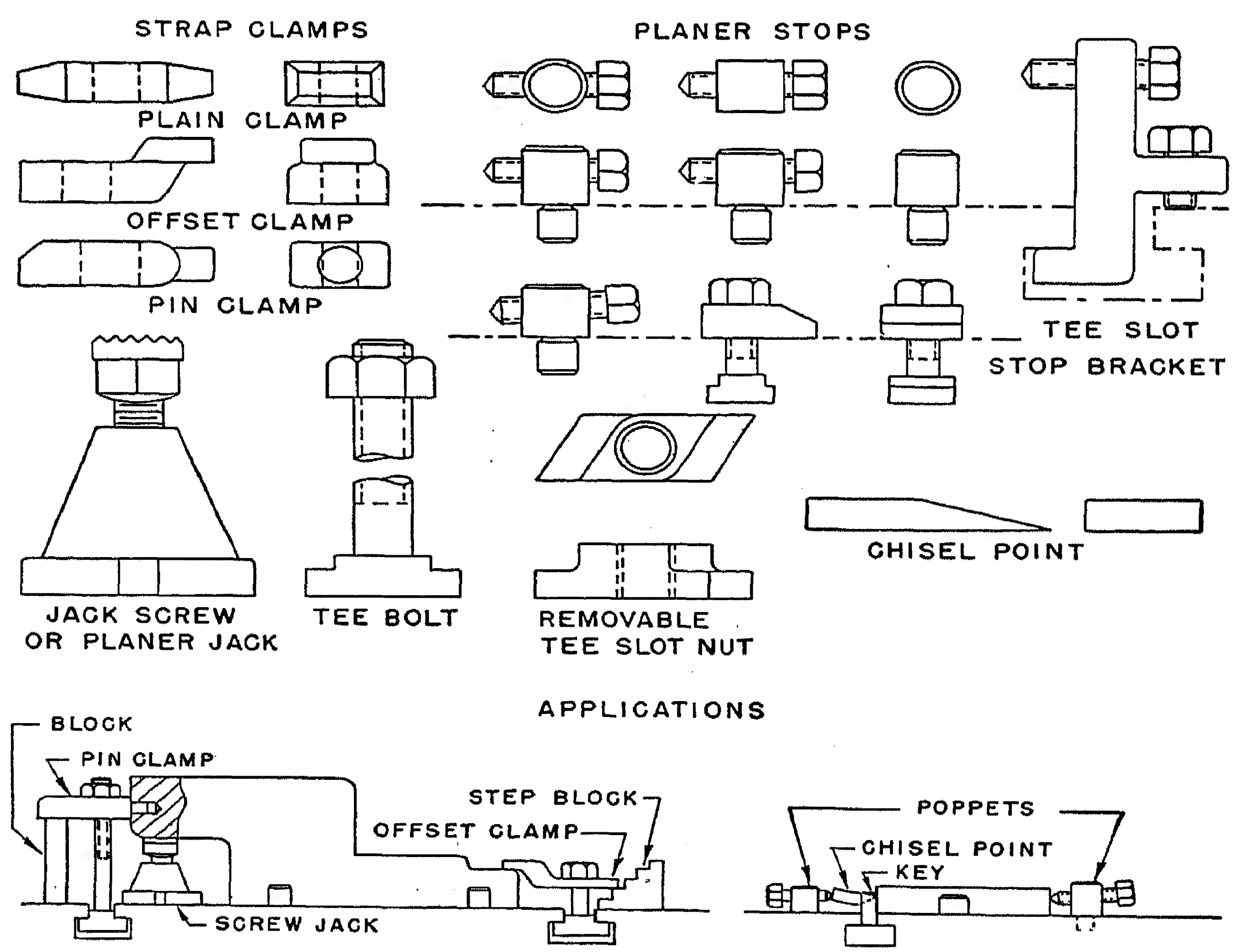


Fig. 10-8. Common devices for clamping and supporting work on a planer.

shank. *Double cutting tools* are arranged for planing on both strokes of the table.

Work-holding devices. Most workpieces are clamped directly to the table on a planer. Common devices for clamping and supporting workpieces are shown in Fig. 10-8. Strap clamps are the most positive holding means. They are fastened with T bolts and must always bear at both ends. The offset clamp allows the clamping nut to be set down out of the way. A pin clamp has a projection to enter a hole or recess in the side of the workpiece. Planer stops are placed in the table holes or T slots to locate and clamp the work. Chisel points are backed up by clamping screws and are inclined at 8° to 12° to hold down a piece on which the whole top surface is to be machined. Jack screws provide support for overhanging sections. When a T bolt with a square head is used in a T slot, the chips must be cleaned out of the slot to the end of the table if the bolt is to be inserted or removed. That does not have to be done for removable T slot nuts and bolts, because they can be slipped in from the top of the table at any position and are locked in place when turned in the slot. A workpiece may be clamped to an angle plate to plane one side square with another.

Sometimes work is held in a vise or vises fastened to the top of the table. Fixtures are used when justified by the quantity of pieces to be machined. Setup time can often be saved by duplicate sets of fixtures. One fixture is unloaded and loaded off the planer while parts in the other fixture are being machined on the planer. *Setup plates* have T slots and holes on top like a planer table and keys in the bottom to fit the center T slot of the planer table. One setup plate is loaded on trestles next to the machine while another is on the planer table with work being machined. The setup plates can be taken off and put on the planer table more quickly than can the workpieces.

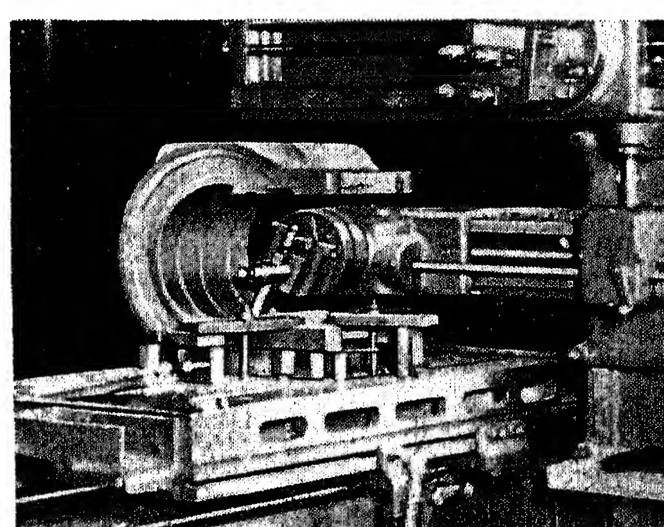


Fig. 10-9. A circular planing head for railroad driving boxes. (Courtesy Rockford Machine Tool Co.)

Tool setting gages. A gage

or template is often placed on top of a planer table to set a tool for roughing or semifinishing cuts. A standard planer gage applied in Fig. 3-23 is commonly used for general-purpose work. It can be adjusted to any height in its range. Various gages are made for specific jobs. They are commonly incorporated in planer fixtures. A tool is set by bringing it down to touch a piece of paper on top of the gage so that the paper can just be pulled from between the tool and gage.

Surface generating attachments. A number of attachments has been developed for planers to generate accurate external and internal round surfaces. A typical attachment of this type is shown in Fig. 10-9.

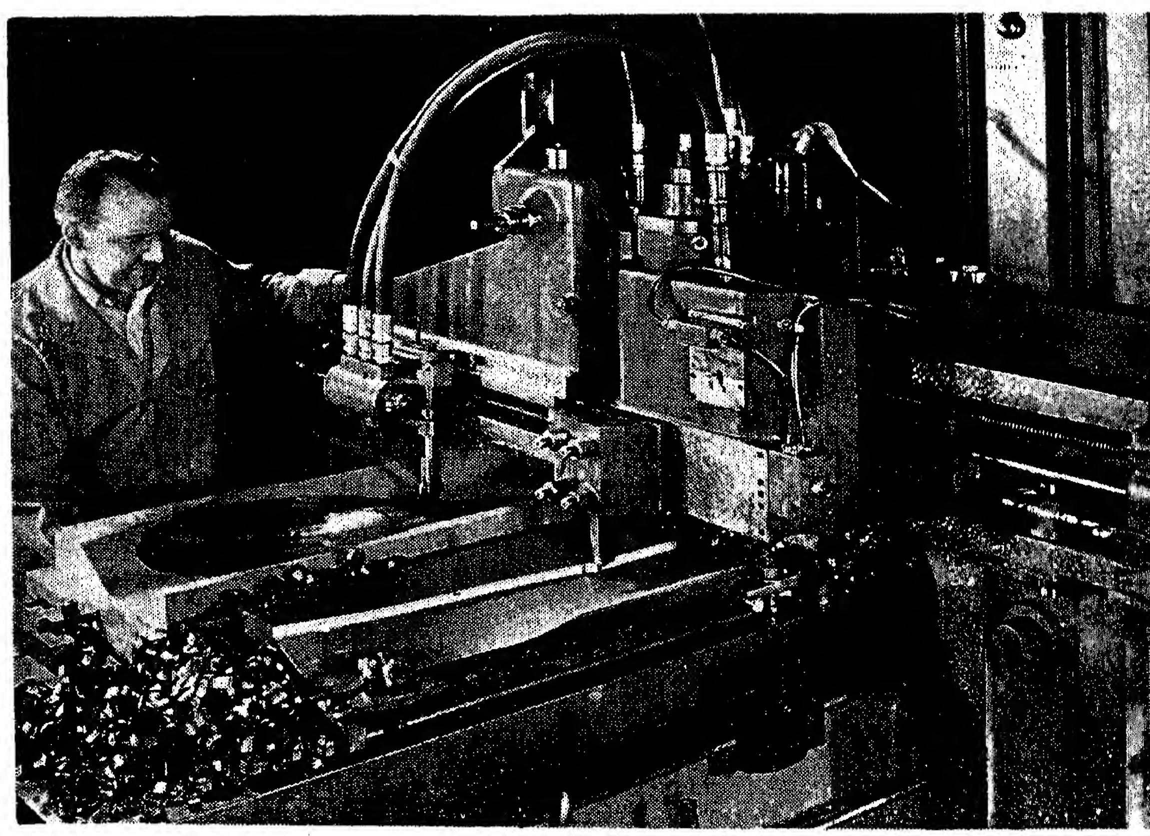


Fig. 10-10. A hydraulic duplicator planing an irregular form. (Courtesy Rockford Machine Tool Co.)

Many sizes and shapes of surfaces may be reproduced accurately by planing through the use of hydraulic duplicating attachments. An example is given by Fig. 10-10. A special rail head carries a tracer finger and cutting tool. During each stroke the finger follows the contour of the master on the table, and the tool is guided to reproduce the profile on the workpiece.

Planer Operations

Setup. Efficient setup practice is probably the most important phase of planer operation. It has been said that a planer is more difficult to set up than any other machine tool, in spite of the fact that it is a relatively simple machine in itself. Planers are capable of taking heavy cuts, and large forces must be expected. Cutting forces also are often intermittent, as when a number of small or medium-size parts are machined at one time. Inertia forces from table reversal must be taken into account. The effect of cutting and clamping forces in distorting and bending the work must also be considered.

A plentiful supply of clamping and holding details should be available and used freely to hold the work securely. Stops are placed at the ends of a workpiece whenever possible to keep it from shifting along the table. Paper is commonly placed below the clamp points to increase friction and protect the table. When both sides of a piece, particularly a plate, are to be machined, the piece often has to be turned over several times. First one surface is planed, then the other side, and so on to neutralize the strains and reduce warping.

A workpiece must be positioned properly on a planer table if it is to be machined accurately. Work is often laid out for planing. Lines are scribed to show the positions of the finished surfaces. The work is then positioned so that the surfaces to be cut in a setup are aligned with the machine movements. Horizontal lines are checked for parallelism with the table top by a surface gage and for squareness by a square. Wedges, shims, and paper are put under the work where needed to hold it in the correct position.

To get the most out of a planer, as many tools as possible must be used for each job. The tool heads provide three or four tool stations. Often a number of workpieces can be placed side by side or in two rows to utilize the heads fully.

Tool overhang should be avoided on a planer the same as on a lathe or shaper. The heads should be kept as close to the work as possible and the slides should not be overextended. A tool should not extend beyond the clamping bar more than is necessary to avoid interference from the tool head.

The large work commonly done on planers is not expected to be

machined as closely as smaller work done on other machines, such as shapers. Tolerances of 0.005 to 0.010 in. are considered practicable, although smaller tolerances are realizable with care and skill.

Planing compared with other operations. Planers generally are capable of performing the same basic operations as shapers and milling machines. Because their basic members are well supported, planers can be made in large sizes to take bulky and heavy pieces or a number of pieces. Although the planer is not as suitable as the shaper for relatively small and medium-size parts done one or a few at a time, it often is faster and more economical where larger quantities are required. The components of a planer are massive and rigid, and heavy cuts and feeds are feasible.

Planing with single point tools is generally conceded to be a slower cutting operation than milling with multiple point tools. However, the planer has a definite place in metal machining because both the machine and the tools are relatively low in first cost and upkeep. As a rule, the planer offers more machine capacity at a lower cost than the milling machine. If a part can be planed almost as fast as it can be milled, a planer may be more desirable because of the low cost of grinding the tool bits. For a typical operation, enough planer tool bits could be ground in about an hour for a 24 hour operation. Several hours would be spent in grinding milling cutters for the same operation.

Estimating planing power and time. A planer tool is usually set to rough a surface to about 0.005 in. above size. The depth of cut depends upon the amount of stock to be removed and sometimes is as much as one inch. For fair finishes, the remaining stock is removed by the finishing tool. For fine finishes, three cuts are customary. From 0.001 to 0.002 in. of stock is taken off by the final cut. Broad nose finishing tools for cast iron are fed $\frac{1}{4}$ to 1 in. per stroke at speeds of 30 to 40 sfpm for high speed steel and about 200 sfpm for carbides. Feeds up to $\frac{1}{4}$ in. per stroke are recommended for finishing steel.

The limitations of the tools, work, and machine determine the speeds and feeds that can be used for rough planing. Planing tools usually can be run at about the same surface speeds as turning and shaping tools. The speed and feed often are limited by the size of the work that must be reversed at each stroke, by the clamping

that can be applied to the work, or by the weakness of the workpiece itself. The machine speeds and power available may be limiting factors, the latter especially with several tools operating at once. No values can be given for speeds and feeds to cover all cases. The speeds suggested by Table II in Chapter 4 may be used as a guide but must be modified by conditions. Feeds generally range from 1/32 to 1/8 in. per stroke. Lower speed is desirable with heavy feed.

The number of strokes per minute depends upon the length of stroke and the time required for the cutting stroke, return stroke, and reversal. These factors vary with the machine and operation. Average strokes per minute that have been recommended for estimating time on medium-size modern planers are:

Length of stroke, in.	8	12	18	27	40	60
Number of strokes per min	46	37	28	20	14	12

The cutting time is calculated from the number of strokes per minute, the feed in inches per stroke, and the distance the tool is fed in inches, in the same way as described for the shaper.

The horsepower per cubic inch per minute of stock removal may be comparable to that required on a shaper for light cuts. Metal removal is more efficient for heavy cuts usually taken on a planer. One manufacturer of planers uses the constants of 0.25 hp per cu in. per min for cast iron, 0.5 for machine steel, and 0.15 for bronze.

Questions

1. Describe a standard planer and an openside planer.
2. How is the size of a planer designated?
3. How are modern mechanical planers driven?
4. Describe the drive system of a hydraulic planer.
5. Name the major units of a standard planer and tell what purpose each serves.
6. How are planer tools like shaper tools? How do they differ?
7. How may work be held to a planer table?
8. What is the purpose of a tool setting gage?
9. How may curved or irregular surfaces be generated on a planer?
10. Why is a planer difficult to set up?
11. How much should a planer tool be allowed to overhang?
12. When may a planer be the most suitable machine tool for an operation?

Problem

Estimate the cutting time to rough and finish plane in 2 passes 4 sides of cast iron blocks of the following dimensions on a 42 in. by 42 in. by 10 ft mechanical planer with a 35 hp motor and a gear ratio of 5. One quarter inch of stock is to be removed from each surface with high speed steel tools.

(a) 6 in. x 12 in. x 18 in.
(b) 12 in. x 18 in. x 24 in.
(c) 24 in. x 42 in. x 4 ft
(d) 18 in. x 36 in. x 5 ft

References

Manufacturers' Catalogs:
Baldwin-Lima-Hamilton Co., Philadelphia, Pa.
Cincinnati Planer Division, Giddings and Lewis Machine Tool Co., The, Fond Du Lac, Wisc.
Consolidated Machine Tool Corp., Rochester, N.Y.
G. A. Gray Co., Cincinnati, Ohio.
Rockford Machine Tool Co., Rockford, Ill.

Chapter 11

DRILLING MACHINES AND OPERATIONS

DRILLING MACHINES ARE USED mostly for opening, enlarging, and finishing holes by drilling, boring, counterboring, countersinking, reaming, tapping, and spotfacing. These operations, variations of them, and less common operations performed on drilling machines will be described in this chapter.

Types of Drilling Machines

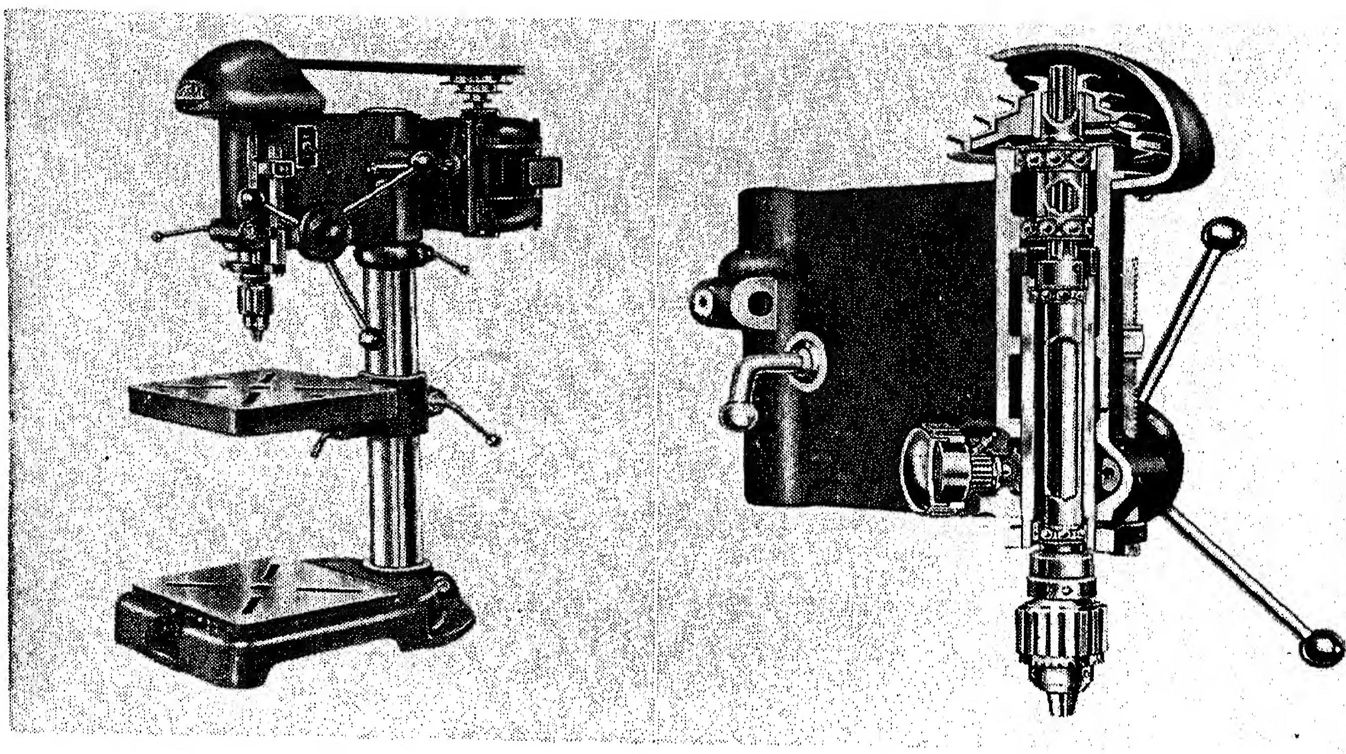
Drilling machines are made in many forms and sizes. Portable or hand drills in models ranging from simple ones cranked by hand to large electric and air-driven drills are well known. They are widely used for general utility, repair, and maintenance work in manufacturing plants, on construction projects, around garages, and even in homes. Portable drills are sometimes mounted on upright brackets or bench stands to serve as stationary drilling machines.

The drilling machines commonly used for precision metal working are known as drill presses. They have means for holding or supporting the work and a mechanism for forcefully feeding the spindle that rotates the cutting tool. A drill press may have one or a number of spindles, vertical or horizontal spindles, hand or power feed, be heavy or light, large or small, or possess any of a number of other features. The chief types of drill presses and their characteristics are described in the sections that follow.

Bench-type drill presses. Bench-type drill presses are small machines for light work. A typical one is shown in Fig. 11-1. Although short in stature, it has many of the basic features of larger drill

presses. A column on a base supports a table for the work and a head that carries the drill spindle. The table can be adjusted up or down, swung around the column, tilted around a horizontal axis, and clamped in position. Slots in the table are for bolts to fasten the work. The table can be swung out of the way so that work can be mounted on the base, which also is slotted.

A cross-section of the head of the drill press is included in Fig. 11-1. The spindle revolves in a pair of ball bearings in a nonrotating



A B

Fig. 11-1. A 15 in. bench type drill press and a cross-section of the head showing the spindle construction. (Courtesy Atlas Press Co.)

sleeve, called a quill. The quill is fitted and moves up and down in the head. Rack teeth along the quill mesh with a pinion on the shaft that carries the hand spokes on the side of the head. As one of the spokes is pulled forward and down by hand, the quill and spindle are lowered together. A coil spring on the left end of the cross shaft counterbalances the spindle and causes it to return to its highest position when the hand lever is released. This construction is typical of many drill presses of all kinds and sizes.

A motor is adjustably mounted on the rear of the head and drives a stepped pulley around the spindle. The hub of this pulley revolves in a pair of ball bearings in the head and has internal

splines to engage the splines on the upper portion of the spindle. As the spindle moves up and down, it slides in the pulley but is rotated by the pulley at all positions. The drill press of Fig. 11-1 has a 1725 rpm constant speed motor and 9 spindle speeds from 600 to 5200 rpm.

The spindle of a bench-type drill press often has a drill chuck pressed on its lower end but it may have instead a small tapered hole. A stop is carried on the quill to set a limit to its descent. A clamp on the front of the head locks the quill if desired. The entire head can be lowered or raised, swung around, and clamped in position on the column. Some heads of this type are counterbalanced.

Light drill presses that are hand fed so that the operator can feel the resistance met by the drill are called *sensitive drill presses.* They are advantageous for feeding small drills to avoid breakage. Many have spindle speed ranges up to 5000 rpm and over.

Very small holes must be drilled on ultrasensitive presses with spindles running very true at high speeds. One such type of drill press has the spindle rotating in a pair of V blocks instead of conventional bearings. Microscopes are used to watch the action of the drills. Holes less than 0.001 in. in diameter have been drilled, and diameters around 0.005 in. are common in the instrument and watch industries. Drills below 0.012 in. diameter are flat instead of helical because they are stronger and can be sharpened more easily.

Upright drill presses. The drill press in Fig. 11-1 is also made with a long column so that the base can be set on the floor instead of on a bench. A heavier floor-type machine is the general-purpose *standard upright drill press* of Fig. 11-2. It has a box-type column that is more rigid but does not provide the adjustments that a round column does. Many upright drill presses have round columns.

The table in Fig. 11-2 is mounted on ways on the column, is raised or lowered manually by an elevating screw, and can be clamped to the column for more rigidity. The table has slots for work clamping bolts and a cutting fluid trough around its edges. Some machines have a compound table on a saddle for adjustments in two horizontal directions. Others have round tables that can be swiveled.

The spindle quill of the drill press of Fig. 11-2 is carried on a counterweighted sliding head that is raised and lowered by a hand crank and clamped to the front of the column. The quill may be

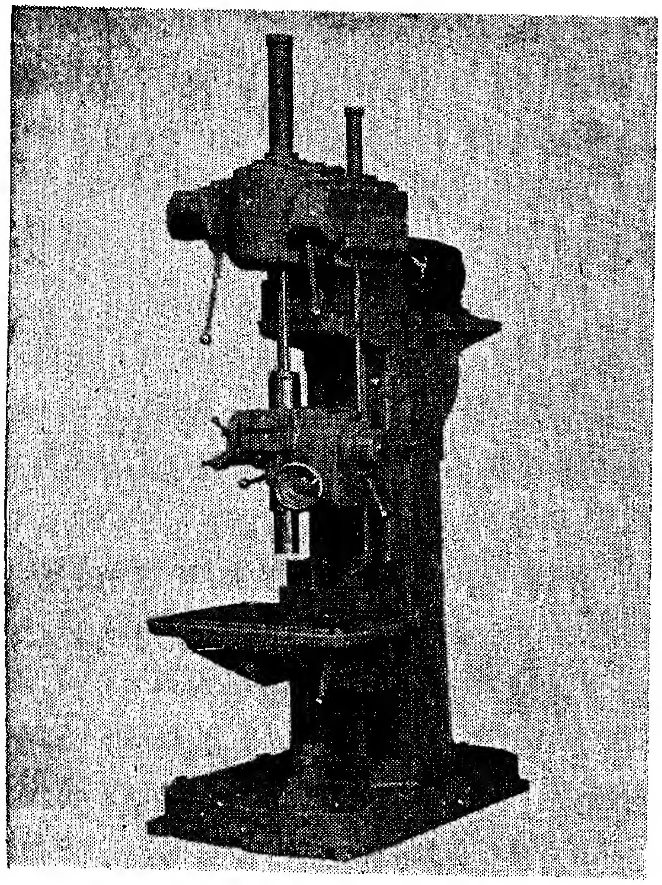


Fig. 11-2. A 25 in. single spindle upright drill press. (Courtesy The Fosdick Machine Tool Co.)

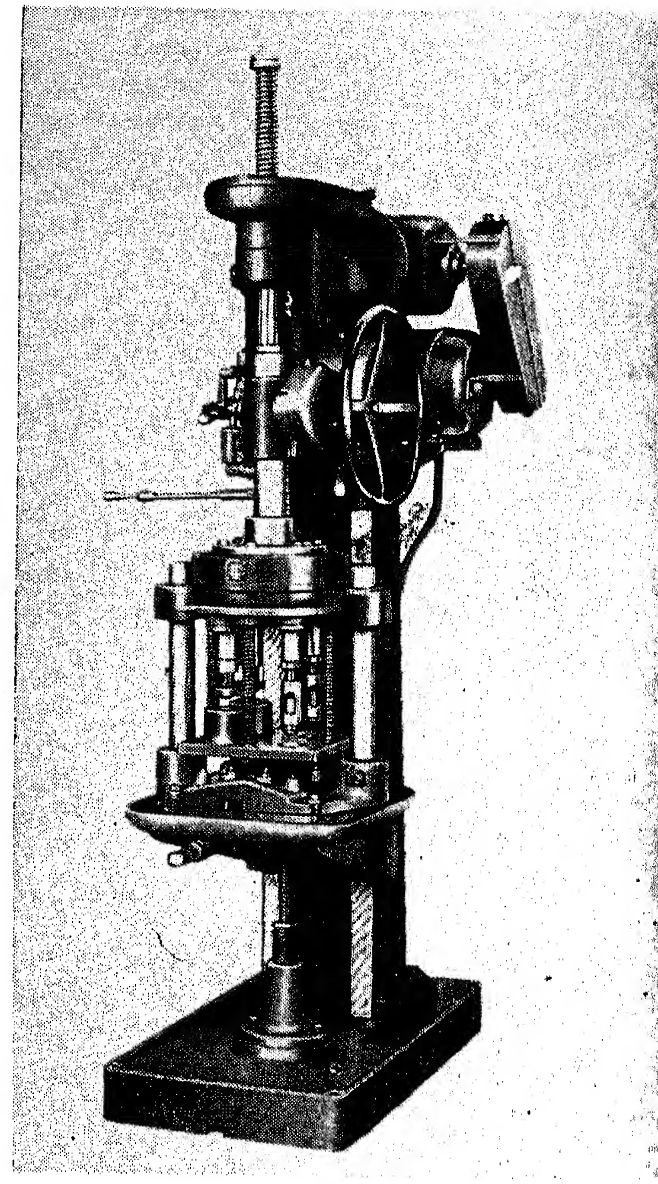


Fig. 11-3. A single spindle production drilling machine with a four spindle auxiliary head and jig with pressure plate. This machine has special tooling for counterboring, countersinking, and tapping two spark plug holes in a cast iron cylinder head cover. (Courtesy Barnes Drill Co.)

fed by power or by hand. Power for the feed drive is brought down from a feed box on top of the column by a spline shaft. This machine has 9 power feed rates from 0.005 to 0.043 in. per revolution. Machines like this may be equipped with a positive leadscrew for tapping and a spindle reversing mechanism. On some, an adjustable stop is arranged to limit the depth of travel of the quill, to disengage the power feed at a definite depth, or to reverse the spindle rotation while tapping. The quill and spindle are raised by a counterweight or spring. The spindle has a No. 4 Morse taper hole. Generally the larger the machine, the larger the tapered hole.

Modern drill presses have individual motor drives, and the power

often is applied to the spindle through a change gear box, as is the case with the machine of Fig. 11-2. The motor is on a ledge on the rear of the column and drives the transmission in the box on top of the column. The spindle drive gears are shifted by the lever at the top left. Each speed is designated on a chart by a pointer on the shifting lever. This machine has 12 speeds in geometrical progression from 60 to 1500 rpm. The upper part of the spindle that extends into the gear box is splined and driven in any position. Power for the feed drive is transmitted from the spindle drive to the feed box on the right on top of the column. Feeds are selected by means of the shift lever on the right and are indicated on a chart by a pointer.

Some drill presses have motors wound for several speeds and mounted directly around the top of the spindle, so that only a few or no gears are necessary.

Production drill presses. Production drill presses are sturdily built for heavy work and simple in design. They are intended for long runs on specific jobs and are not easy to change from one job to another. They often are equipped with special tooling and attachments to enable them to do certain jobs most efficiently.

A typical production drill press is illustrated in Fig. 11-3 with a capacity to drive $\frac{3}{8}$ to 1 in. high speed steel drills in mild steel. A 3 hp motor drives a gear box in the head through a silent chain or V belt. A cover on top of the head must be removed to change pick-off gears in order to change speeds. Pick-off gears are available for speeds from 69 to 1296 rpm, but only a set for one speed is furnished as standard equipment. A cover plate on the side of the head is taken off to change pick-off gears for feeds from 0.0035 to 0.030 in. per revolution.

Special tooling and attachments have been added to the basic machine in Fig. 11-3 for machining a part in large quantities. A jig with a bushing plate is mounted on the table. An auxiliary 4 spindle head is bolted to the flange on the end of the quill. This machine has a reversing multiple disk clutch and brake in the gear box for automatically reversing the spindle for tapping. Feed gears are installed to give a positive lead equal to the pitch of the taps. Machines like this may also be equipped with dwell attachments, indexing tables with multiple stations, and raising blocks. A *dwell attachment* provides a pause at the end of the spindle travel to

permit a tool, such as a spotfacer, to clean up a surface. An *indexing table* with two or more stations enables an operator to load work in one station while cutting is done at others. *Raising blocks* are placed under the head or column to accommodate tall work.

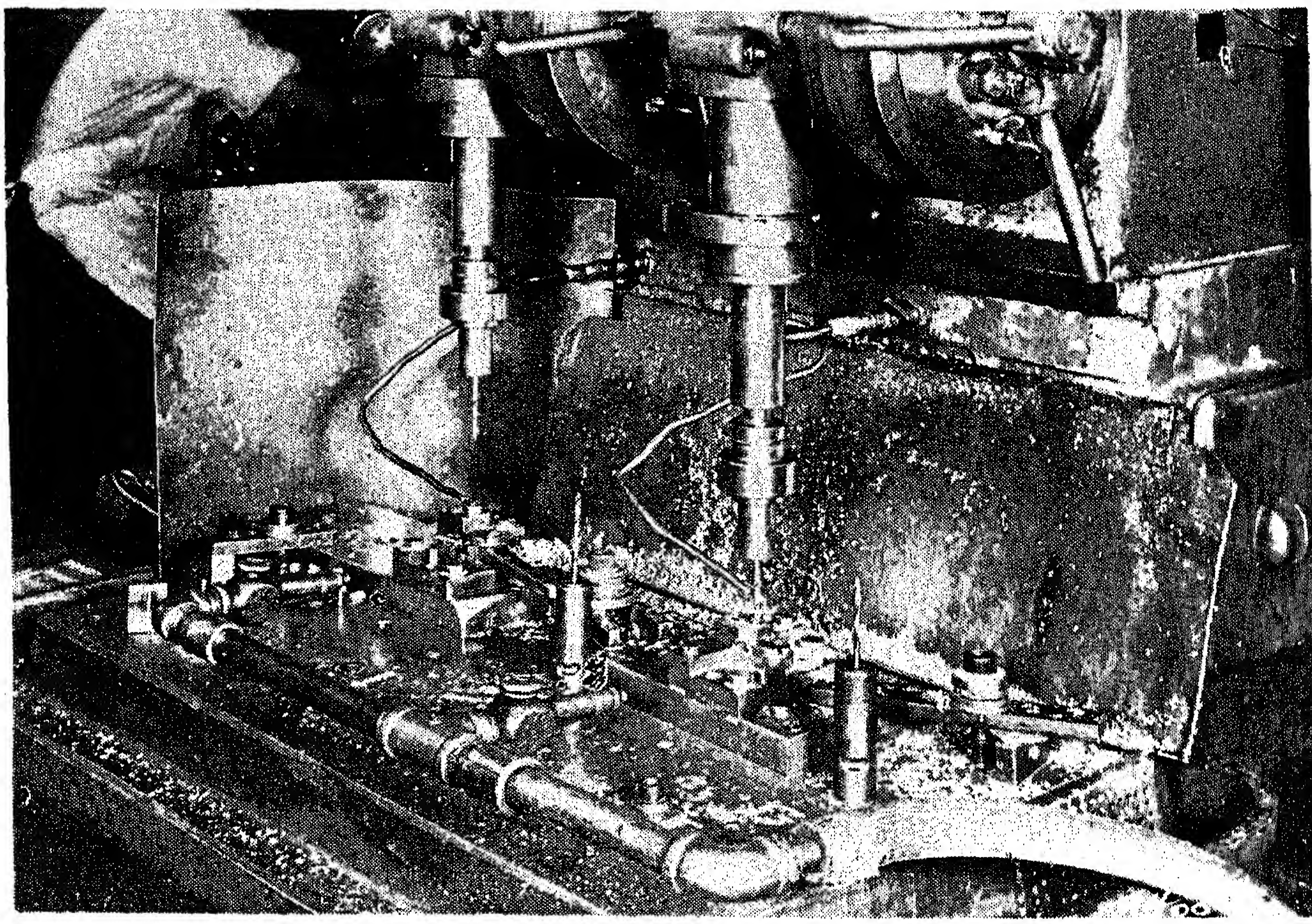


Fig. 11-4. A production job installed on a two spindle gang drill press. A jig is clamped to the table by strap clamps under each spindle. Magic quick change chucks are mounted on the spindles, and the tools are held in quick change collets. An air line is arranged with a nozzle for blowing the chips away from each jig. (Courtesy Consolidated Machine Tool Corp.)

Gang drill presses. A gang drill press has two or more independent spindles on separate heads in a row. The heads may be like those on general-purpose upright drill presses or like those on production drill presses. A production job mounted on the table under two spindles of a gang drill press is shown in Fig. 11-4. Some gang drills have a separate table for each spindle, and each table can be adjusted separately for the work done on it. Each spindle of a gang drill has its own drive and can be run at speeds and feeds independently of the other spindles.

Gang drills are helpful in production for doing several operations on a part or for machining several parts at one time. To do several operations, a different kind of tool such as a drill, boring cutter, reamer, or tap may be put in each spindle and run at the speed and feed best for it. The workpieces can be moved from spindle to spindle by one operator or be passed along by several operators, one at each spindle.

Several parts may be efficiently machined at one time on a gang drill when the loading time is small compared to the cutting time. In that case, all the stations are equipped with identical tooling. An operator unloads and loads one jig while the tools at the other stations are working. He then proceeds to attend to another station, and so on.

Sizes of drill presses. The size of a single spindle or gang drill press designates the diameter in inches of the largest disk in which a center hole can be drilled on the press. The size is nominally twice the distance from the center of the spindle to the column. Thus, a 21 in. drill press is one that is able to drill a hole in the center of a piece 21 in. in diameter. The ordinary range of sizes is from about 6 to 50 in.

Multiple spindle drill presses. A multiple spindle drill press has a cluster of spindles on one or more heads for a part having a number of holes and produced in large quantities. The spindles may all do the same kind of work or different kinds like drilling, boring, reaming, and tapping. Standard drill presses are made into multiple spindle presses by adding auxiliary heads to them, as in Fig. 11-3, but many multiple spindle drill presses are constructed with integral heads. They range in size from small ma-

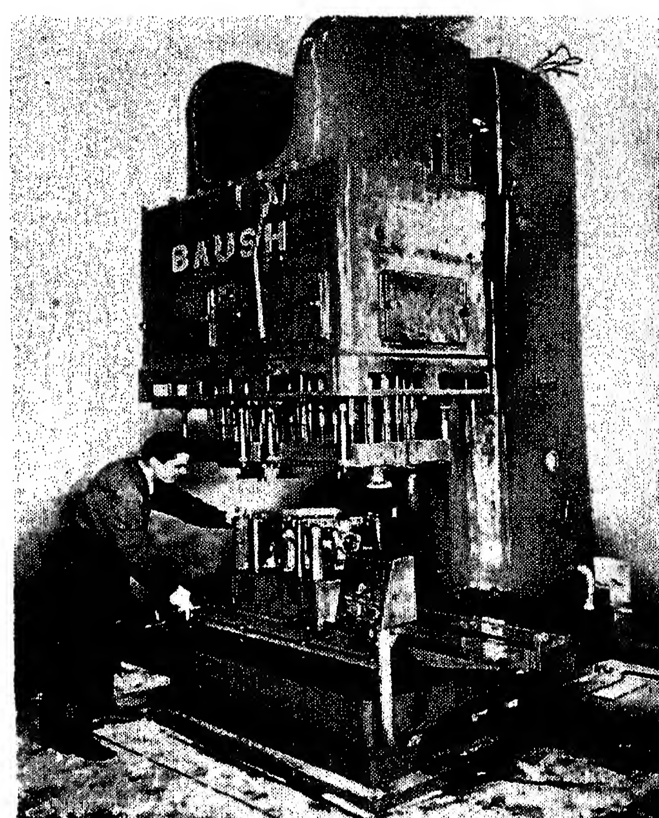


Fig. 11-5. A multiple spindle drilling machine for drilling 50 holes in a gear case. (Courtesy Bausch Machine Tool Co.)

chines powered by a few horsepower to massive ones driven by as much as 50 hp. Some of the more elaborate of these have heads and spindles in more than one position to operate upon the tops, sides, and angular surfaces of parts at the same time or in successive stations. They are usually equipped with special jigs and fixtures and often have indexing tables with a loading and one or more machining stations.

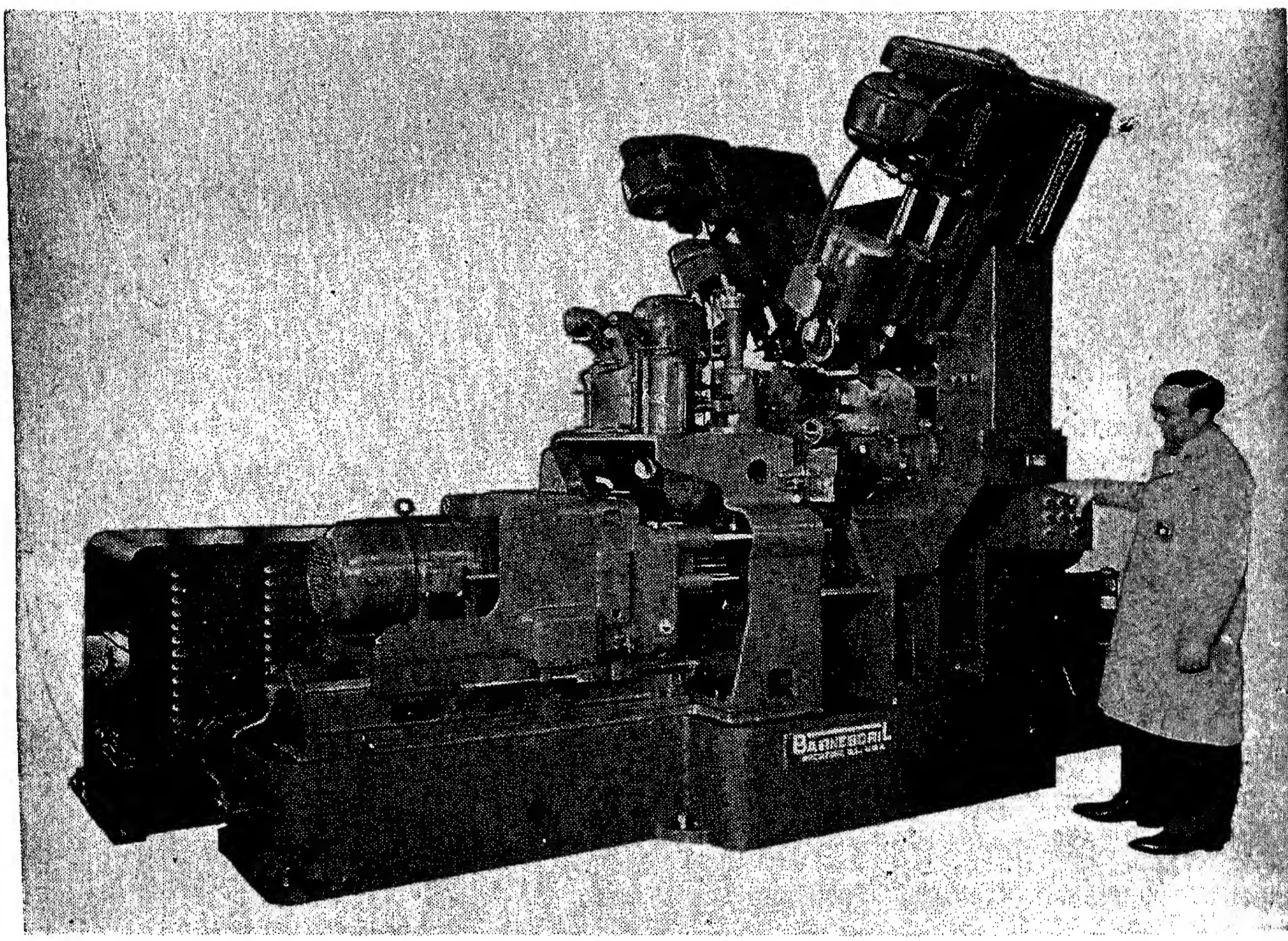


Fig. 11-6. A special multiple spindle high production drilling machine with six heads. (Courtesy Barnes Drill Co.)

A large multiple spindle drilling machine for drilling 50 holes in the front end of a gear case of an aircraft engine is illustrated in Fig. 11-5. The 22 by 45 in. head of the machine is fed hydraulically, with a rate of feed almost infinitely adjustable between ¼ and 8¾ in. per minute. The spindles are supported in and located by a fixed center precision bored cluster plate. The drills are guided in a bushing plate suspended from the head. The part is held in a two-station manually indexed fixture. The operating cycle of the

machine is semiautomatic. A 25 hp motor drives the spindles, and a 3 hp motor the hydraulic unit. Twenty parts with a total of 1000 holes are produced per hour.

Some multiple spindle drilling machines have a stationary drill head fastened to the top of the column. The work table below the head is raised and lowered hydraulically to feed the workpiece to and retract it from the tools.

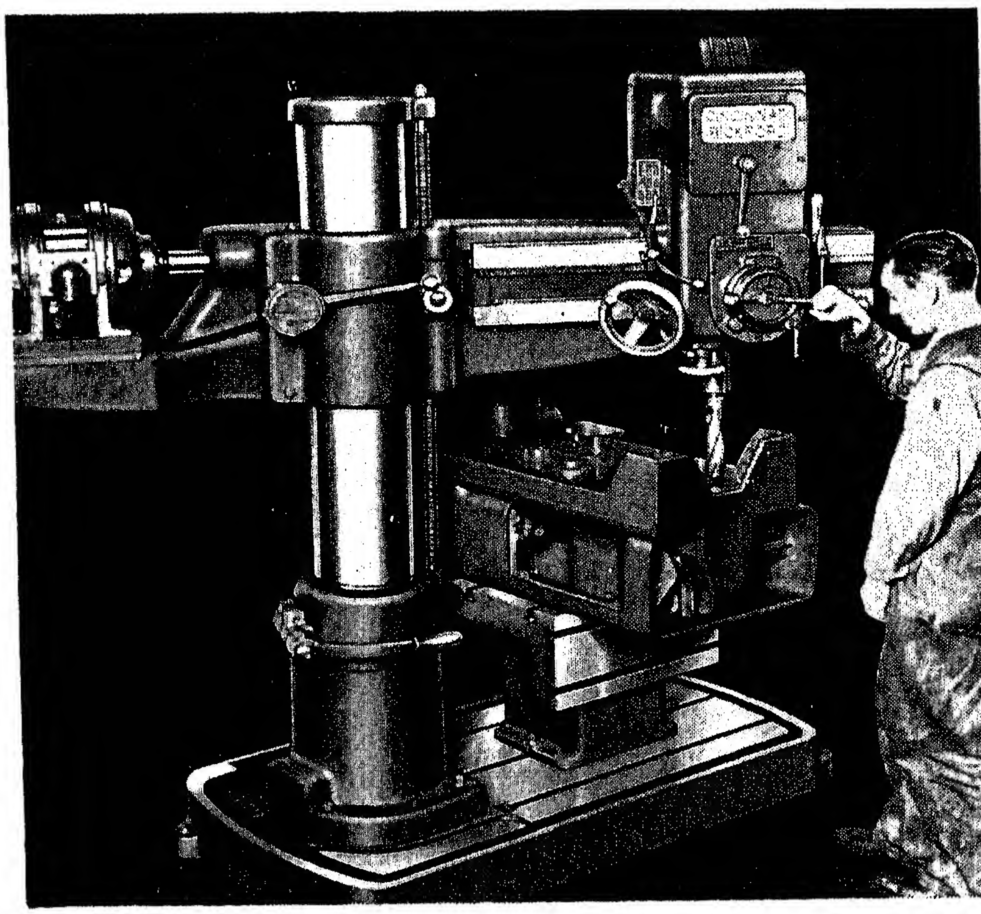


Fig. 11-7. A 4 ft radial drill. (Courtesy Cincinnati Bickford Tool Co.)

An example of a highly developed multiple spindle production drilling machine is given by Fig. 11-6. That machine drills, chamfers, reams, faces, and taps holes in a cast iron automobile transmission case. The heads have individual motors for spindle drives but are fed, and the multiple station fixture is indexed, hydraulically. A workpiece is indexed through seven stations and is acted upon by six multiple spindle heads.

Radial drills. Radial drills are convenient for heavy workpieces that cannot be moved around easily or are too large for other kinds

of drill presses. A *plain radial drill press* has a base with a vertical column that carries an arm supporting the spindle head, as shown in Fig. 11-7. The arm can be raised or lowered by power on the column and clamped in position. The column swivels to swing the head in a circular path and also can be clamped to prevent rotation. The head can be adjusted along the arm by hand and clamped in position. The drill spindle is fed up and down by a quill in the head by hand or power. The drive motor on one end of the arm helps counterbalance the head. Nine speeds and four feed rates are available. A choice of one of six speed ranges is given when the machine is purchased. The lowest is from 60 to 1200 rpm and the highest from 175 to 3500 rpm. A feed range of 0.002 to 0.010 in. per revolution or 0.004 to 0.020 in. per revolution may be selected.

The size of a radial drill press is given in feet and designates the radius of the largest disk in which a center hole can be drilled when the head is at its outermost position on the arm. This size corresponds to the distance from the column to the center of the spindle. Thus, a 4 foot radial drill is capable of putting a hole in the center of a piece having a radius of 4 feet. Some makers specify the diameter of the column in inches in addition to the radial capacity in feet.

Radial drills vary in size from those with a 9 in. diameter column and 3 ft arm, like the one in Fig. 11-7, to a 26 in. diameter column and 12 ft arm. The small size may have a 3 hp motor and is intended for drills up to about 1 in. diameter. Small radial drills sometimes are termed *sensitive* because they are designed for light high speed drilling, mostly with hand feed. The biggest radial drills may have as large as 40 hp motors and be capable of drilling and boring holes as much as 16 in. in diameter. The large sizes have more speeds and feeds than small sizes because they must be able to satisfy more requirements. A typical large radial drill has 36 spindle speeds from 12 to 1200 rpm (or as low as 6 to 600 rpm if ordered) and 18 feeds from 0.006 to 0.125 in. per revolution. These include tap leads for 8, 10, 11½, and 14 threads per inch.

A *universal radial drill press* has all the movements of a plain radial drill press, and in addition its arm can be rotated 180° about its own horizontal axis and its head may be swiveled in a plane parallel to the face of the arm. The spindle can thus be positioned at any angle over the working area.

The base of a radial drill like the one in Fig. 11-7 has T slots, and large workpieces often are bolted to it. Work-holding fixtures and jigs may also be mounted on the base as shown in Fig. 11-8. The trunnion and outboard support for that jig are typical of standard work-indexing accessories.

Various kinds of tables are available for radial drills. A *plain box table* supports the workpiece in Fig. 11-7. It has two working surfaces square with each other and containing T slots. A *universal table* has a top part like a plain table, and the top can be tilted on a pedestal. It is used to hold work at a desired angle. Some universal

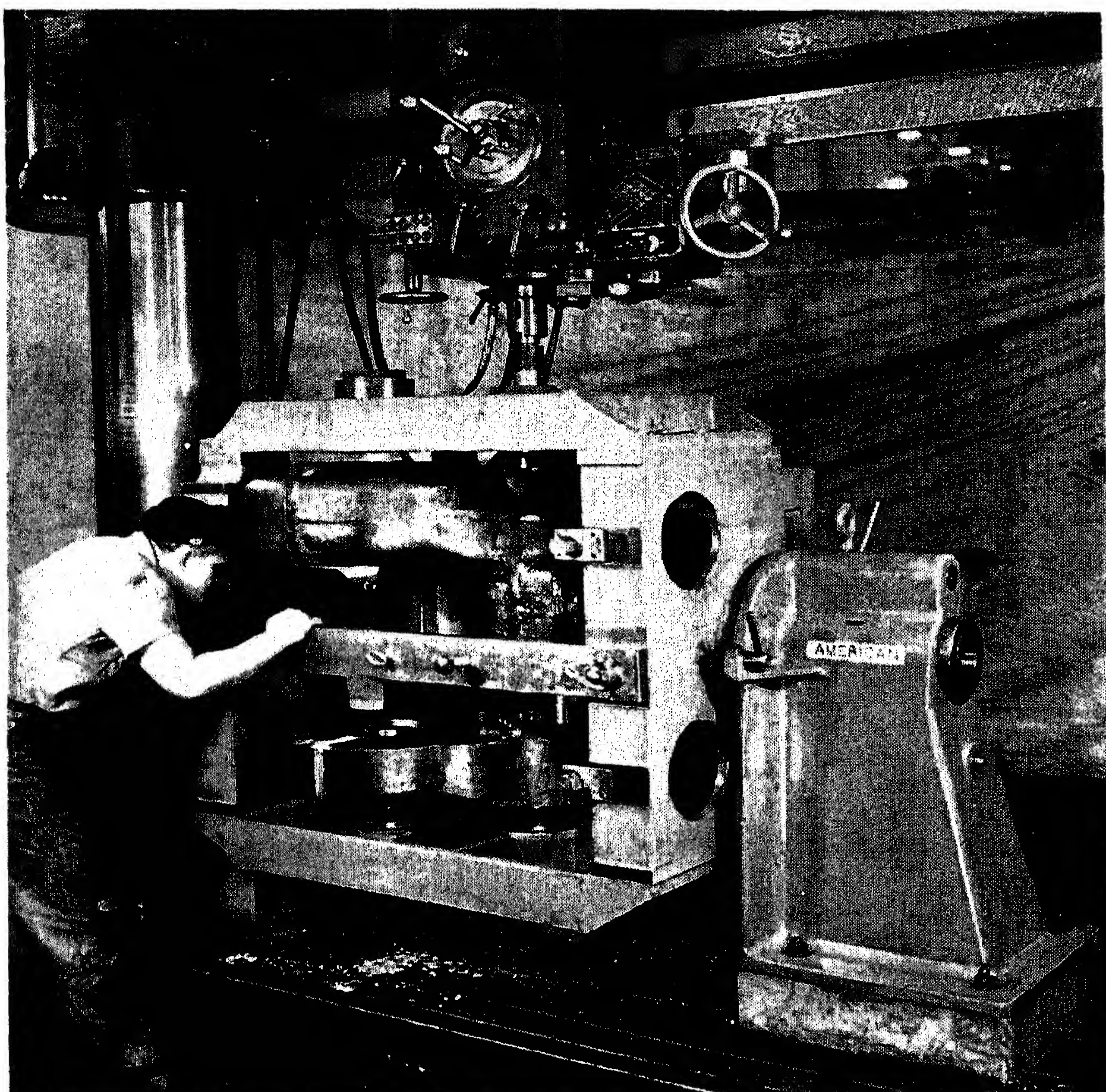


Fig. 11-8. A trunion jig on a radial drill press for drilling and boring a heavy transmission frame. (Courtesy American Tool Works Co.)

tables also swivel in a horizontal plane. A *swinging table* is supported at one end on and can swing around the column. It may be supported at the other end by a foot that stands on the base of the machine. The swinging table has a finished top and side with T slots and is almost as long as the base. In effect, it acts like an auxiliary base that can be swung out of the way. Some swinging tables can also be tilted. A *round table* has radial T slots and can be revolved as desired.

The radial drill base shown in Fig. 11-7 is the most common, but some have an extension at right angles. Other types of bases are available to make a radial drill portable. A *track-type base* is not intended to carry the work but has wheels that travel on rails. The arm and head extend beyond the base to take care of work to which the radial drill is brought. The drill press may be moved by hand or power and fastened in any position on the tracks. A radial drill with a *sliding base* is arranged so that the column on its base can be moved by power along ways of a bed to various positions. That allows the drill press to cover a large working area, 15 feet long in a typical case. The machine can operate on a workpiece in one place while others are being readied for it.

Deep hole drilling machines. The drilling of deep holes presents several problems. Chips are difficult to clear out of a hole when a drill is cutting at a depth many times its diameter. When a drill is burrowing deeply into metal, its point is hard to reach with a fluid to cool and lubricate it. The runout of a drill may become excessive in a long hole. These problems must be solved in drilling centrally such parts as gun barrels, crankshafts, camshafts, and hollow spindles.

Deep holes that do not have to run true or have a smooth finish are drilled with standard two lip twist drills. Either the work alone is revolved or both the work and drill are revolved, because rotation of the work tends to influence the drill to follow the axis of the workpiece. A step feed is used. That means that the drill is withdrawn from the hole each time it is fed a distance equal to its diameter. That clears the chips from the hole and makes it unnecessary to flute the drill for the entire length of the hole. With a step feed, a drill having two diameters may be used where necessary. If the hole is over ¾ in. in diameter, a drill with oil holes between the flutes is used, and oil at high pressure is supplied to the point.

That helps wash out the chips and supply cutting fluid where it is effective.

A *crankshaft drill* is a special twist drill for deep holes smaller than ⅜ in. diameter, such as oil holes in crankshafts. It has a high helix angle, a thick web, and a point ground with a 55° secondary clearance angle on the heel to thin the web at the point and break up chips.

Deep holes requiring smooth finish and minimum runout are drilled with single flute deep hole *gun drills* like that illustrated in Fig. 11-9. These are operated at high speeds (about 130 sfpm for HSS) and low feeds (0.003 to 0.006 in. per revolution) as com-

Fig. 11-9. Two views of a typical single flute deep hole gun drill. (Courtesy National Automatic Tool Co., Inc.)

pared with twist drills. Only the work is revolved. Oil is supplied through a hole running the length of the drill and washes the chips back through the straight flute. Deep holes may be reamed with standard reamers after drilling to improve finish. Only the reamer is revolved, and the feed is continuous.

Standard upright and production drill presses sometimes are modified for deep hole drilling. The drill may be mounted in an upright position in a socket on the table or base. Oil is supplied under pressure to the socket for oil hole drills. A fixture carrying the workpiece is attached to and revolved by the spindle from the head and fed to the drill. The power feed mechanism is arranged to provide step feed.

Horizontal drilling machines like the one in Fig. 11-10 are made specifically for deep hole drilling. The head on the right of that machine has a quick speed change gear driven spindle and a hydraulic feed arranged for either step or continuous drilling. That head feeds the drill and revolves it if desired. The left-hand head

carries the work which can either be locked stationary or revolved. When revolved, the spindle is driven by an electronically controlled variable speed motor adjustable through a wide range of speeds.

Deep hole drilling machines are made with a number of spindles for large-quantity production. Some are horizontal, others vertical. They are designed to revolve the work, provide step feeds, and in other ways meet the requirements for deep hole drilling.

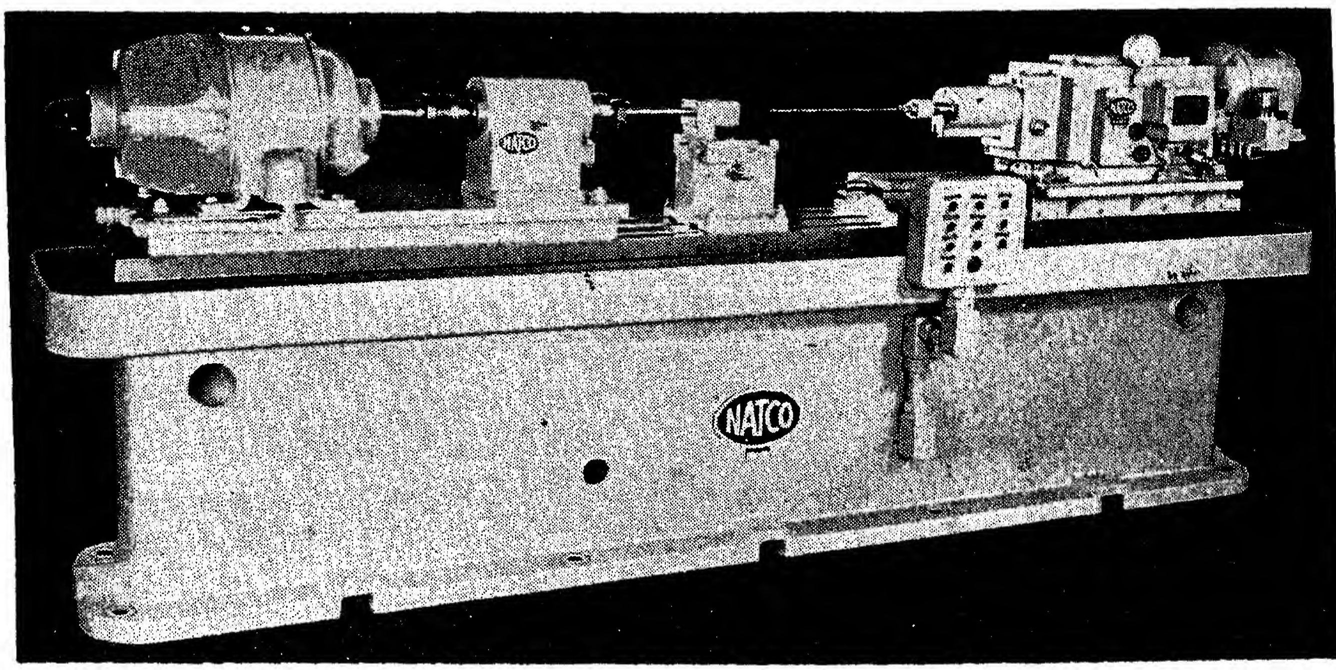


Fig. 11-10. A deep hole drilling machine. (Courtesy National Automatic Tool Co., Inc.)

Tools and Attachments

Cutting tools. The principal cutting tools used in drilling machines are twist drills, centerdrills, and boring, spotfacing, countersinking, and counterboring tools. These tools are also used on lathes and were introduced in Chapter 6 in connection with lathe operations. Taps and dies also are used often on drilling machines. They have been described in Chapter 7. The student should review the descriptions of those tools at this time.

Multicut drills make holes of two or more diameters and are widely used for production work on drilling machines. Typical multicut drills and work that can be done with various forms are depicted in Fig. 11-11. A *step drill* is shown at the top. It is a standard drill that is ground to a smaller diameter on the end. The margin of the

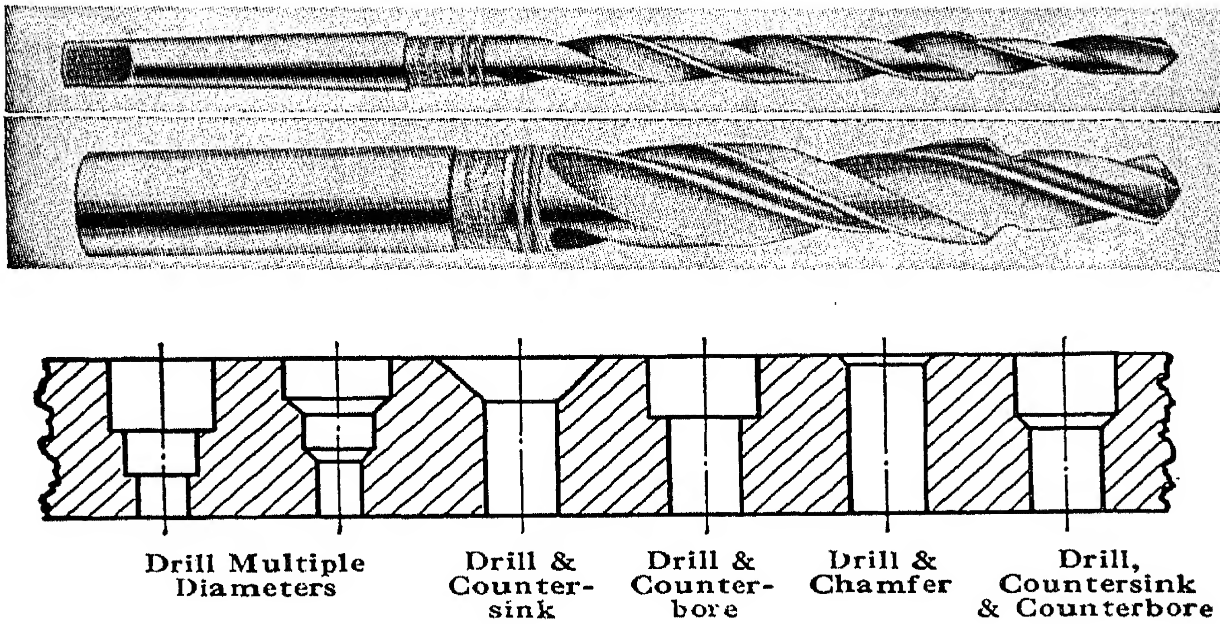


Fig. 11-11. Multicut drills and the work they do. (Courtesy Chicago-Latrobe Twist Drill Works.)

small diameter must be undercut where the drill size changes to produce a sharp corner. After the point has been sharpened back to this undercut, the drill must be entirely reworked or discarded. The *subland type* in the middle view has separate margins for each diameter and costs more but can be used up entirely without loss, and its net cost per hole is less.

Multicut drills with as many as four diameters have been successful, but generally the largest diameter should not be more than twice the smallest because of the difference in cutting speeds. Small diameters less than 3/16 in. are not recommended.

Short diameters, like the outside of a boss, can be turned on a drilling machine by means of a *hollow mill.* One is illustrated in Fig. 11-12. Hollow mills are also used on turret lathes and automatic screw machines. The blades are pitched and sharpened so that they cut on their inner corners and front edges. The work enters the hollow space in the body of the tool and is withdrawn when the cut is finished.

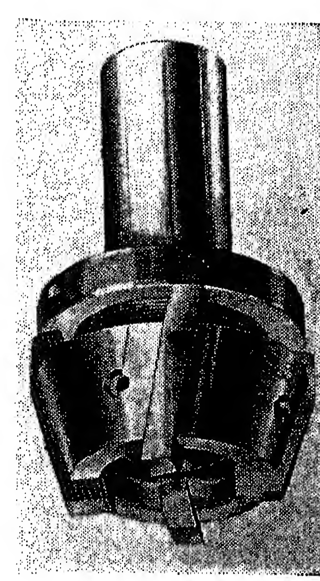


Fig. 11-12. An adjustable blade hollow mill. (Courtesy Gairing Tool Co.)

Drill performance. A drill is subject to two

main forces. One is the *thrust force* in an axial direction required to feed the drill into the material. The other is the *torque force* needed to revolve the drill. Practically all the power used in drilling is related to the torque because the velocity of rotation is so much larger than the axial feed.

General recommendations for drilling, boring, and reaming speeds and feeds were given in Chapter 6. An increase in either speed or feed shortens drill life. Often the speed can be lowered and the feed raised to increase the rate of metal removal without loss of tool life, up to the feed the drill can stand.

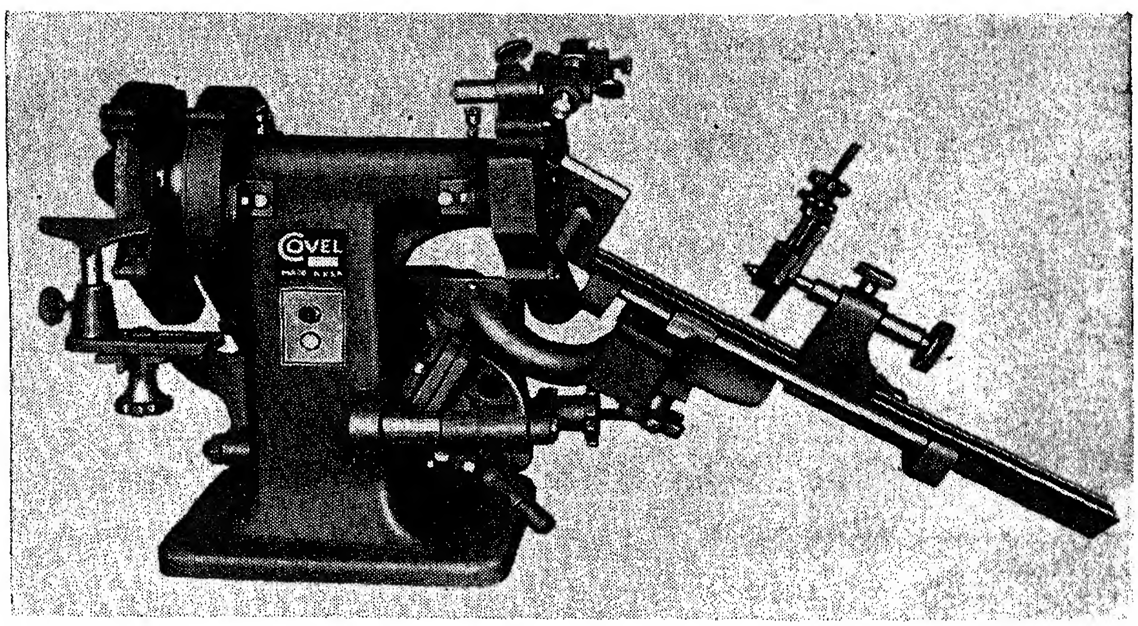


Fig. 11-13. A twist drill grinder. (Courtesy Covel Manufacturing Co.)

Drill grinding machines. The need for the proper angles and cutting edge lengths on a drill is explained in Chapter 6 and illustrated in Fig. 6-13. Machine ground drills generally cut faster, last longer, and produce more accurate holes than drills sharpened off hand. A number of grinders are made for the sole purpose of drill grinding. In principle, they are designed to hold a drill correctly and swing it across a grinding wheel to grind the proper angles uniformly on the point.

A typical drill grinder is shown in Fig. 11-13. An arm on the right of the machine carries a V to support and locate the point of the drill. An adjustable tailstock takes the shank end of the drill. Two sizes of V supports accommodate drills from No. 30 (0.1285 in. diam.) to 1½ in. diameter. A drill is inclined for the usual point

angle of 118°. Adjustments can be made on some machines to produce other point angles if desired. The arm with the drill on it is swung around an axis to grind the heel of the drill. Adjustment is provided for the desired clearance angle. After one side has been ground, the drill is turned over, and the other side is positioned in the same way and ground. A wheel on the other side of the machine can be used for general off hand grinding or for point thinning.

Some drill grinders produce a relief angle back of the cutting edge that is larger at the chisel point than at the outside corner. This conforms to the helix angles in the path of the cutting edge as the drill is fed.

Boring tools, reamers, and taps are held between centers or in collets when sharpened, and the sharpening often is done on a tool and cutter grinder. Machines of that type are described in Chapter 17. Some specialized machines are available for grinding the flutes and tapers of taps.

Toolholders and tool drivers. Straight shank drills, reamers, etc. may be held by a drill chuck illustrated in Fig. 5-13. Various other kinds of drivers are available for straight shank tools. One is a split collet with a straight hole that just takes the tool shank. The outside of the collet has a Morse taper that fits into a sleeve, adapter, or the hole of the drill spindle. When the collet sleeve is pushed in place, it is contracted and grips the tool shank.

Straight shank taps with square tangs are held and driven in a number of positive ways. One tap driver is like a small two jaw chuck. Others include square hole sockets and split collets.

A tool with a tapered shank may be placed directly in the tapered hole of the spindle of a drilling machine. If the tapered shank is smaller than the tapered hole, a taper shank socket, like those in Fig. 6-20, may be used. When a reamer or tap needs to follow a previously drilled or bored hole, a *floating driver* is used. It has two parts. One is inserted in the spindle; the other holds the tool. The two parts are fastened together so that one drives the other, but the connection is free enough to allow the tool to float.

A variety of tapered sockets or sleeves is characterized by a flat on one side of the tapered hole. They are known as *"use-em-up"* sleeves. A flat is ground on the side of a tool shank to allow it to fit such a sleeve. The tool is located from the partial taper but driven

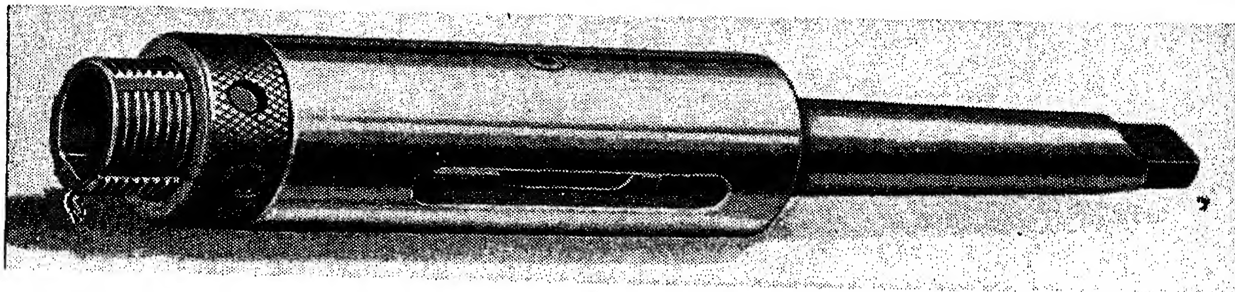


Fig. 11-14. An adjustable extension assembly with a Use-Em-Up hole. (Courtesy Scully Jones and Co.)

by the flat. Drills that have broken tangs can be salvaged in this way. The adjustable extension assembly of Fig. 11-14 has a "use-em-up" hole.

Drills, taps, etc. of different lengths are often found on one multiple spindle head. They must be arranged to extend various distances from the head. Adjustable extension assemblies like the one in Fig. 11-14 are used for that purpose. The inner sleeve has a tapered hole to take the tool and is held in place by a set screw in the outer body. An adjusting nut positions the inner sleeve.

A *quick change chuck* allows tools to be changed on a drill press while the spindle is running. *Magic quick change chucks* are shown in Fig. 11-4. A tapered shank on the chuck

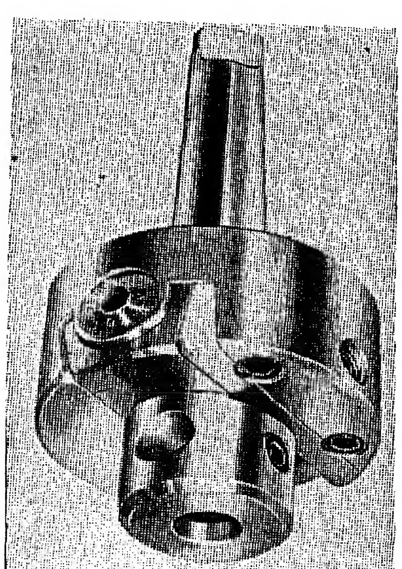


Fig. 11-15. An adjustable offset boring head. (Courtesy Flynn Manufacturing Co.)

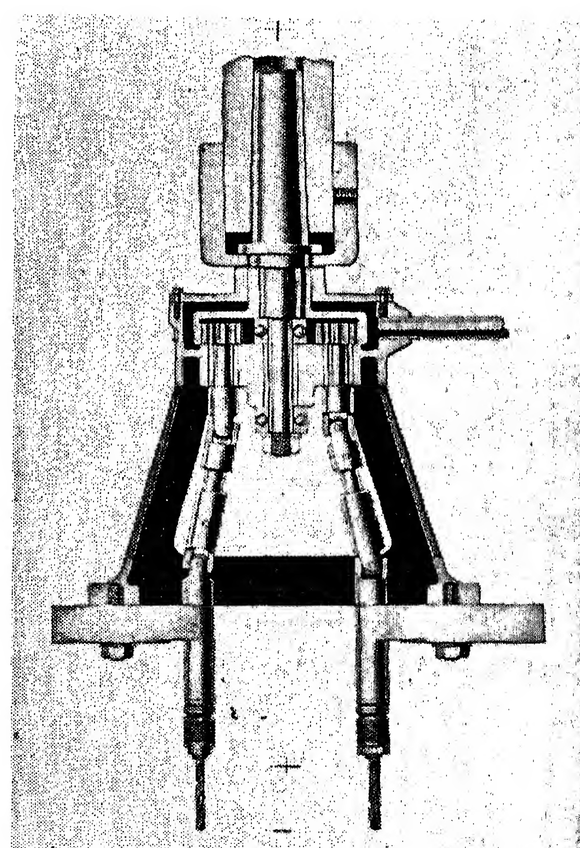


Fig. 11-16. A cross-section of a portable and adjustable multiple spindle drill head. (Courtesy Baush Machine Tool Co.)

body fits the drill spindle. A round collet slips easily into the hole in the chuck and is automatically secured by a latch. A ring around the chuck body is lifted to release the latch and allow the collet to be withdrawn. A variety of collets is available to accommodate drills, reamers, boring bars, taps, etc. A number of operations can be done on a piece with this chuck from a single spindle without loss of time for stopping the spindle for changing tools.

Adjustable boring heads, like the one in Fig. 11-15, are used on drilling, boring, and milling machines. The tapered shank of the head is inserted in the machine spindle. The block that carries the boring tool is closely fitted in the body of the head and is adjusted off center by a screw and the micrometer dial shown. A diameter within the range of the head can be bored to an accuracy of 0.0002 in.

Multiple spindle drill heads. Auxiliary multiple spindle drill heads are mounted on drilling machine spindles for driving two or more tools at one time. An example of the use of such a multiple spindle head is given in Fig. 11-3. Some of these heads are made for specific purposes. In such a case, the spindles are located accurately and permanently in a precisely bored plate. On other heads, the location of the spindles can be changed. A cross-section of one kind of adjustable multiple spindle head in Fig. 11-16 shows how the spindles are mounted and driven. Heads of this make are built to cover areas from 4 by 4 in. to 12 by 12 in. The smallest size can accommodate 6 spindles, other sizes up to 8, 10, and 12 spindles. The spindles in the head run about four times the speed of the drill press spindle.

Work-holding devices. A workpiece may be clamped to the table

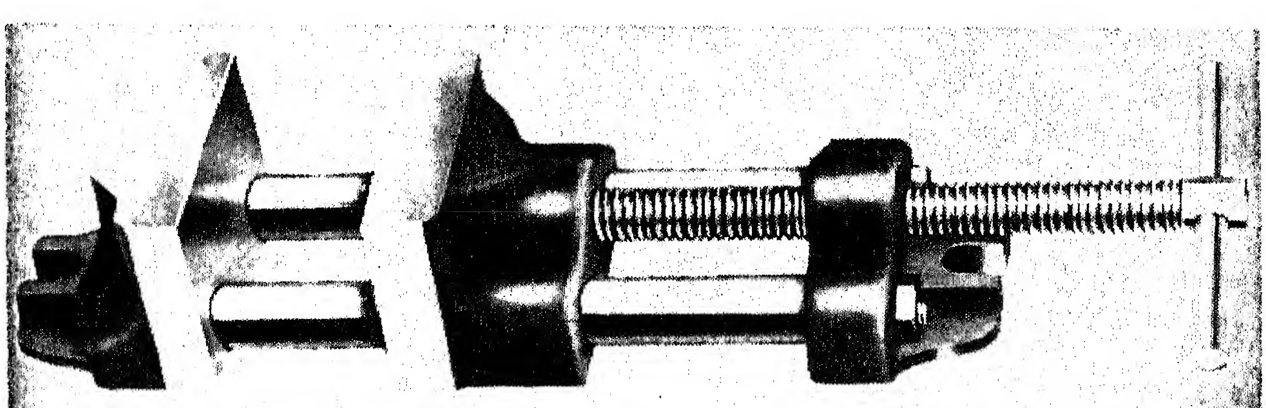


Fig. 11-17. A typical drill vise. (Courtesy Atlas Press Co.)

of a drill press by means of T bolts and strap clamps. Often a workpiece is merely placed on the table and kept from turning by a block or strip bolted to the table. The workpiece is held down by hand, but care must be taken that it is not lifted when the drill breaks through.

V blocks make good supports and locators for round pieces. The work may be held in the block by hand for small drills or clamped in it for larger ones. Workpieces may also be clamped to angle plates. A *universal angle plate* has a T slotted surface that can be tilted and secured at any angle from the horizontal to vertical and can also be swiveled through 360° on its base. Graduated scales show the angles. Some such scales have verniers.

Vises are commonly used to hold work on a drill press. A typical *drill vise* is shown in Fig. 11-17. The jaws are guided by bars and clamped by a screw in the middle. Plenty of space is provided for a drill to pass through a workpiece without penetrating the vise. The jaws are finished on top, bottom, and at least one side so that the vise can be turned 90°. Screw vises are comparatively slow acting. Many vises are actuated by quicker means such as cams, latches, and toggles.

A three jaw chuck, like the ones used on lathes, may be mounted on a base with the jaws up to hold pieces for drilling. The base often contains an indexing mechanism so that holes can be spaced and drilled at equal intervals on a circle.

Fixtures and jigs. A fixture is a device that holds and locates a workpiece. Strictly speaking, conventional work-holding devices such as chucks and vises are fixtures, but mostly the name is given to single-purpose holding devices. Fixtures are mostly associated with lathes, shapers, planers, millers, broaches, and grinders.

A jig not only holds and locates a workpiece but also guides the cutting tool. That feature is particularly desirable in drilling because the point of a drill tends to wander unless it is guided when a hole is started. Multiple point boring tools also need to be guided in a jig on a drill press unless piloted in a previously drilled hole. A reamer may be positioned by a jig for accurate work. However, reamers and taps usually are allowed to follow previously prepared holes while the work is held in a fixture.

The simplest kind of jig is a template that fits a part and has holes to guide one or more drills. More elaborate jigs have locating and

clamping details to locate and hold the workpiece in relation to the jig plate. Jig plates are made of soft steel and have accurately located holes bored in them. Hardened bushings, inserted in the plate to withstand wear, do the actual guiding of the drills, reamers, etc. *Slip* or *removable bushings* are convenient where tools of different sizes are to be used in one hole.

Jigs have many forms. Some of the names descriptive of them are channel, ring, box, open, and index jigs. The machines of Fig. 11-3, 11-4, and 11-8 have jigs mounted on them.

Jigs are helpful in producing duplicate parts economically in quantities. They aid in locating holes accurately and rapidly.

Hole spacer. A hole spacer or *Man-Au-Trol spacer* is a device for positioning work accurately under a drill spindle. A 30 by 20 in. hole spacer is mounted on a radial drill in Fig. 11-18. The size of the work-holding table is 32 × 40 in. The table has a 20 in. movement on a saddle to and from the operator. The saddle can move 30 in. on the base at 90° to the direction of table movement. The table is moved hydraulically to an exact desired location when the operator turns a selector dial to any one of thirteen positions. When the device is set up for a job, stops are adjusted to locate the table where needed for each position of the selector dial. The saddle is located independently in the same way to 20 positions. The table and saddle positioning stops are set initially by accurate length gages and an adjustable micrometer. Once a setup has been made, the positions can be reproduced quickly as many times as desired. Any number of duplicate pieces can be machined by locating them on the table with blocks and other locators.

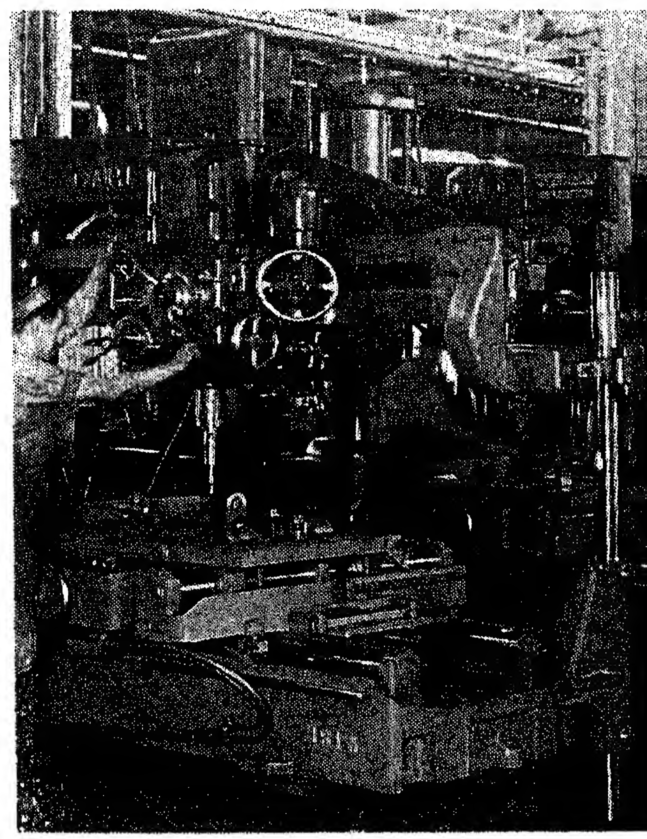


Fig. 11-18. A 30 in. by 20 in. hole spacing device on a radial drill for positioning holes in a workpiece. (Courtesy Bullard Co.)

An outboard support between the hole spacer and radial drill arm in Fig. 11-18 gives added rigidity. A smaller size of hole spacer is available, a 4 by 4 in. unit. It has a 4 by 4 in. stroke and a table 9½ by 11⅝ in.

Drilling Machine Operations

The principal kinds of operations performed on drilling machines have already been named. Only a few remain. One of these is *trepanning*, an operation of making a circular cut to remove a slug, usually from thin material. With suitable tools, drilling machines can be adapted to shaping, mortising, routing, and sanding wood. They are occasionally applied to such metal work as turning, light milling and grinding, spinning of rivets, and polishing but are not as efficient for such purposes as machines specifically designed for them.

Setup and preparation for drill press operations. A drilling machine is relatively easy to set up and operate. For an operation on a single spindle, bench or upright drill press, a drill is chucked or inserted in the spindle, with adapters if necessary. A spindle speed is selected to give the correct surface speed at the outside of the drill. The work is placed in a jig or directly on the table. The table is adjusted to a convenient distance under the drill. The stop attached to the spindle quill is set for the proper depth of hole. Care must be taken in drilling a through hole that the drill is prevented from cutting into the table, vise, etc.

The most elementary way of locating drilled holes in a workpiece is by layout. The methods and tools for laying out work are described in Chapter 3. The face of a block with a layout for 3 holes is sketched in Fig. 11-19. First, centerlines are inscribed, and light indentations made at their intersections with a prick punch. A circle of the diameter of each hole is scribed around the center with dividers. The center mark is deepened with a center punch after the circle has been drawn. That is done afterward because a heavy indentation does not provide a true pivot for the

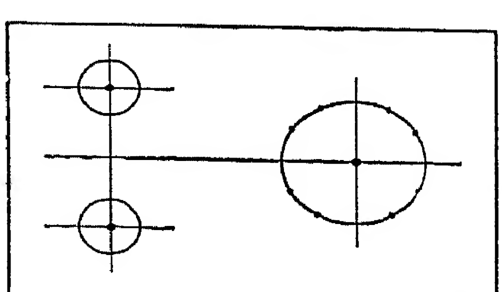


Fig. 11-19. A layout for drilling three holes.

dividers. The locus of the circle may be marked by six or eight light prick punch marks.

The point of a drill is started in the center punch mark. When the point has entered only part way, the drill is withdrawn. The cut made by the drill is usually found to be eccentric with the scribed circle. To correct the situation, a groove is cut in the drilled depression with a round nose chisel on the side toward which the drill should move for correction. That leaves less material for the drill to remove on that side and prompts the drill to shift in the desired direction. Another cut is made with the drill, and the new circle is compared with the scribed circle. If they still are not concentric, the corrective measures may be repeated. When the drill is found cutting true, it is fed to depth.

A drill small enough to be supported by the punch mark should be used to start a hole because the long chisel edge of a large drill overrides the mark. After that, a hole must be enlarged in reasonable steps, because large drills do not follow holes that are too small.

A comparison of drilling machine operations. Even with care, a hole cannot be expected to be located closer than 0.005 to 0.010 in. to a true location by the layout method just described. Obviously, that method is slow, calls for skill, and is suited only for drilling one or a few pieces of one kind, but it requires relatively inexpensive equipment.

A hole spacer can be set up in about the time required to lay out one piece and after that is capable of reproducing the same results quickly. A workpiece is located with less than 0.001 in. error in succesive positions on a hole spacer. The accuracy of hole location depends largely upon how much the drill runs out. A hole spacer is expensive. Although the cost for using it on one job of many is small, it normally would not be found in a shop unless a large amount of work was available to keep it busy.

Holes can be located quite accurately by jigs. Most jigs are usually made for specific jobs and are justified only when moderate or large quantities of parts are produced. Sometimes inexpensive jigs are made to locate holes accurately in one or a few pieces where other means of locating to the desired degree of accuracy are not available or feasible.

The accuracy of hole location is one consideration in planning operations on a drilling machine. Another is the accuracy of hole size

and surface finish. In that respect, drilling, boring, and reaming serve in the same way on a drill press as described for lathe operations in Chapter 6. In summary, a drilled hole is generally not accurate either in size or position. Holes are bored to make them more accurate as well as to enlarge them beyond convenient drill sizes. Reamed holes are accurate in size and have good finishes. Each step adds to the cost of an operation, and those steps must be selected in each case that will give the required results at the lowest cost.

Drilling machine operations compared with others. The operations for which drilling machines are intended can be and are done on other machine tools such as lathes, boring machines, and milling machines. However, drilling machines are best for many jobs. They are simple machines, and their cost is low. Thus, if a job can be done equally well on a drill press and milling machine, the drill press is usually chosen because it represents a smaller investment. Drilling machines also can be tooled for production at relatively low cost, and often a number of operations, such as drilling several holes, can be combined into one.

Because of their simplicity, drilling machines are easy to set up and operate for work for which they are suited. A drill press can be adjusted and changed over quickly from one job to another. Work must be done on a lathe, boring machine, or milling machine around one axis at a time. Adjustments are slow, and appreciable time is required to change from one position to another. In contrast, tools can be changed quickly, work can be shifted around readily, and settings can be altered easily on a drill press to machine holes of different sizes in various positions. For large pieces, a radial drill may be convenient because the work does not have to be moved. The work is usually revolved on a lathe, the tools on a drill press. That is an advantage for the drill press where the work is irregular, hard to hold, or bulky.

A drill press is disadvantageous in some cases. A hole may be machined more economically on a lathe in the same setup with outside surfaces than separately on a drilling machine where the external cuts cannot be taken conveniently. Unless extra equipment like a jig or hole spacer is justified, a drill press is limited in locating holes accurately. More accurate methods of locating holes on milling and boring machines are described in Chapters 13 and 14.

Estimating machining time and power requirements. A hole one in. in diameter is drilled through a cast iron piece 3 in. thick. A HSS drill with a surface speed of 50 sfpm revolves at about 200 rpm. At a feed of 0.015 in. per revolution, the feed rate of the drill is $200 \times 0.015 = 3.0$ in. per minute. The distance the drill is fed is the sum of the length of the hole of 3 in., an overtravel assumed to be $\frac{1}{8}$ in., and an approach of about $\frac{1}{4}$ in.; a total distance of $3\frac{3}{8}$ in. The time for the cut equals $3.375 \div 3.0 = 1.13$ minutes.

The area of the hole is $\pi/4 = 0.785$ in.^{2}. The rate of metal removal is $0.785 \times 3.0 = 2.355$ in.^{3} per minute. For a unit power consumption of 0.8 hp per cu in. per min, the power required is $2.355 \times 0.8 = 1.88$ hp at the tool point.

Questions

1. What is the difference between a bench type and upright drill press?
2. Name and describe the principal units of a typical upright drill press.
3. What are the features of a production drill press?
4. What is a gang drill press?
5. How is the size of a single spindle drill press designated?
6. What is a multiple spindle drill press?
7. Name and describe the principal units of a radial drill press.
8. How is the size of a radial drill press designated?
9. What difficulties must be overcome in drilling deep holes? How are those difficulties overcome in deep hole drilling practice?
10. Describe a typical deep hole drilling machine.
11. What are multicut drills? Describe two types.
12. Why are drills preferably sharpened on drill grinding machines?
13. Describe a typical drill grinding machine.
14. How may work be held on a drill press?
15. What is a fixture? A jig?
16. What is a hole spacer?
17. How may holes be drilled from layouts? With what normal degree of accuracy?
18. What are the advantages and disadvantages of drilling machines as compared with other machine tools capable of doing the same work?

Problems

1. Write an instruction sheet for Operation 6 on the Route Sheet of Fig.

8–9 to drill the 29/64 diam. cross hole in the screw of Fig. 8–8. Only one piece is to be made.

2. Write an instruction sheet for the operation designated in Problem 1 to make 100 pieces. A jig is available for the job.

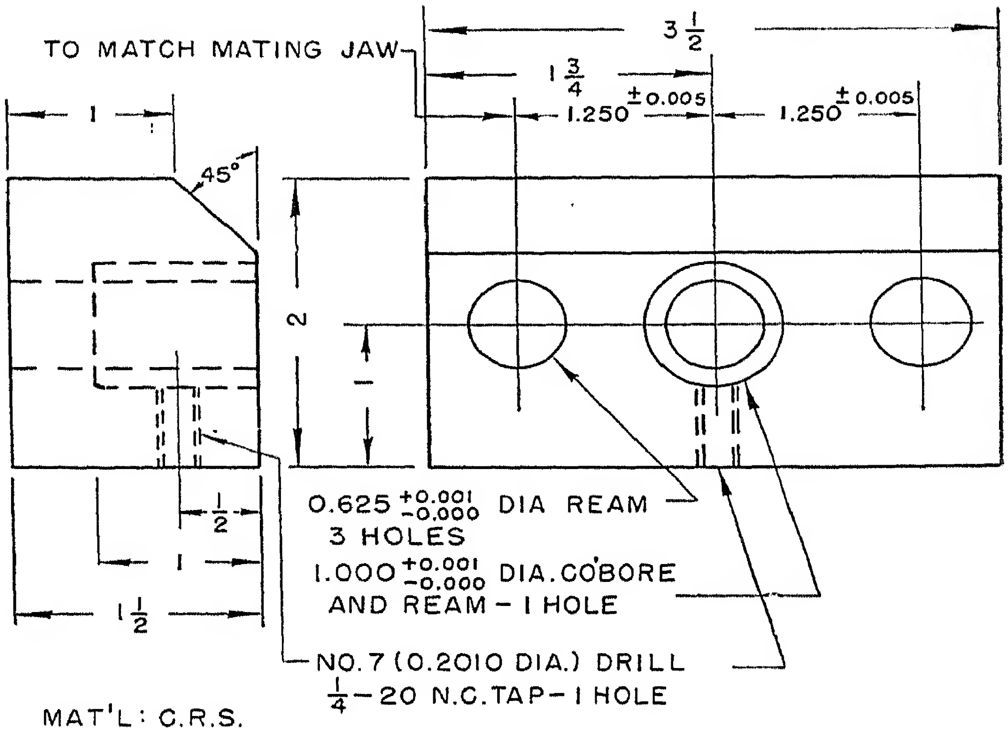


Fig. 11-20. A movable jaw for a small drill vise.

3. Write an instruction sheet for drilling, reaming, and counterboring the holes in the movable vise jaw of Fig. 11-20. One pair of jaws is to be made.

4. Write an instruction sheet for drilling, reaming, and counterboring the holes in the movable vise jaw of Fig. 11–20 for which 100 pieces are required. A jig is available for the job.

References

Manufacturers' Catalogs:

American Tool Works Co., Cincinnati, Ohio.
Atlas Press Co., Kalamazoo, Mich.
Avey Drilling Machine Co., Cincinnati, Ohio.
Barnes Drill Co., Rockford, Ill.
Bausch Machine Tool Co., Springfield, Mass.
Chicago-Latrobe Twist Drill Works, Chicago, Ill.
Cincinnati Bickford Tool Co., Cincinnati, Ohio.
Consolidated Machine Tool Co., Rochester, N.Y.
Covel Manufacturing Co., Benton Harbor, Mich.
Flynn Manufacturing Co., Ferndale, Mich.
Fosdick Machine Tool Co., Cincinnati, Ohio.

Gairing Tool Co., Detroit, Mich.

Maxwell Co., Bedford, Ohio.

National Automatic Tool Co., Inc., Richmond, Ind.

Oliver Instrument Co., Adrian, Mich.

Pratt and Whitney Division, Niles-Bement-Pond Co., W. Hartford, Conn.

Scully Jones and Co., Chicago, Ill.

South Bend Lathe Works, Inc., South Bend, Ind.

Wm. Sellers and Co., Philadelphia, Pa.

Chapter 12

MILLING MACHINES AND CUTTERS

Milling machines are used to machine flat or curved surfaces by means of rotating cutters that usually have a number of teeth. The work is held on a table and fed past or into the cutter.

A milling machine must hold and rotate a cutter and have means to hold a workpiece and move it uniformly in at least one direction. A milling machine must also provide adjustments between workpiece and cutter in three coordinate directions. Its actions must be maintained within definite limits of accuracy.

Many types of milling machines serve in various ways in industry. Some kinds are best for general-purpose work, others are suited for repetitive manufacturing, and some are ideally arranged for special jobs. In general, milling machines may be classified as general-purpose, production, planer-type, and specialized millers.

Types and Sizes

General purpose milling machines. General-purpose or *knee and column milling machines* are among the most versatile of machine tools. They not only do straight milling of flat and irregularly shaped surfaces but also are commonly used for gear and thread cutting, drilling, boring, and slotting operations. They often are applied to production of moderate quantities of parts.

A modern *plain* knee and column milling machine is shown in Fig. 12-1. The column that rises from the base contains the cutter spindle and drive mechanism. The base is hollow and serves to store coolant. The knee slides up and down on ways on the front of the column and is raised by an elevating screw in a telescopic

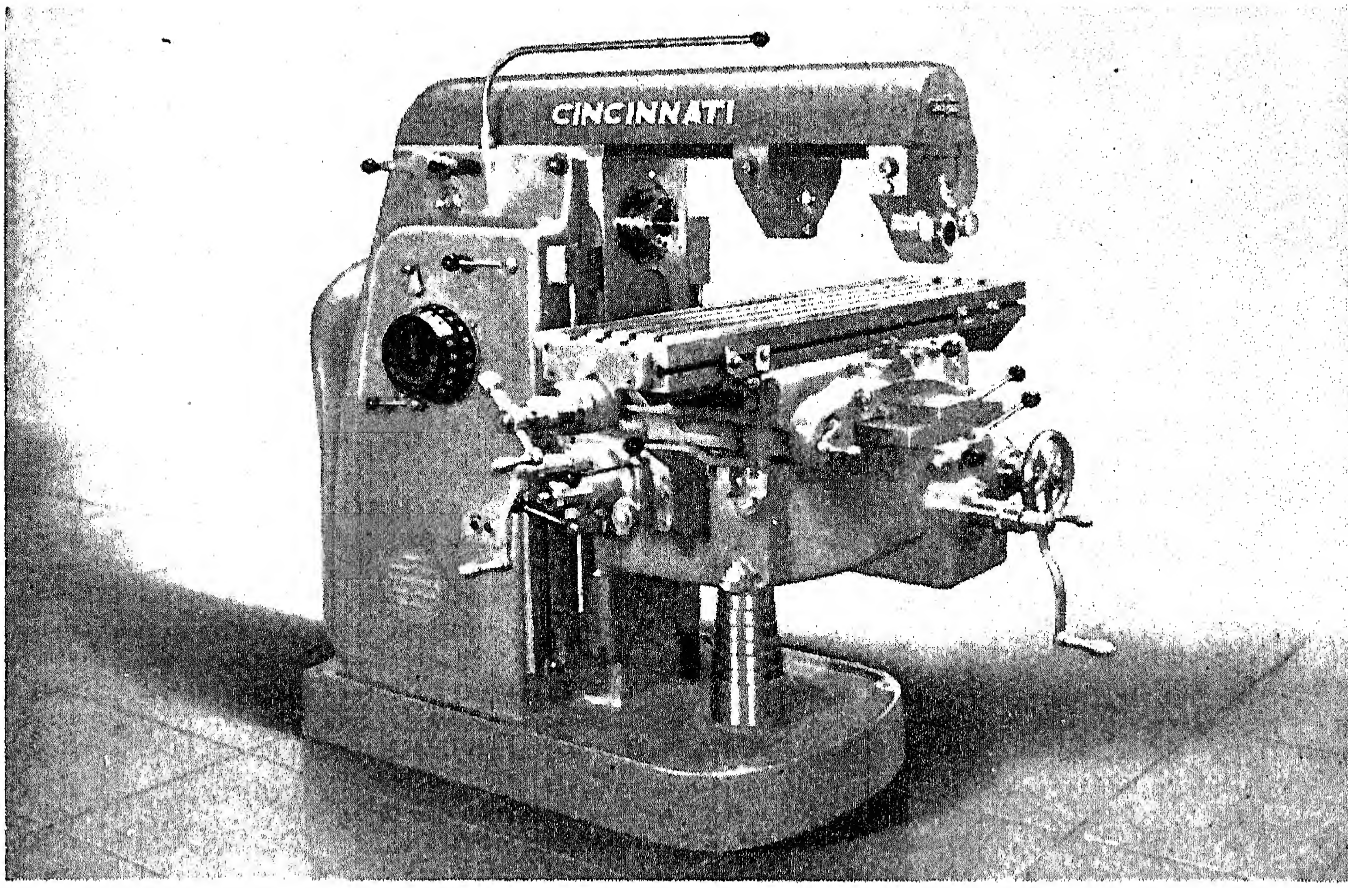


Fig. 12-1. A No. 2 plain high speed horizontal knee and column type milling machine. (Courtesy Cincinnati Milling Machine Co.)

cover on the base. A saddle slides on top of the knee toward or away from the column. The table slides lengthwise on the saddle. The overarm on top of the column carries the arbor supports. Both overarm and arbor supports can be positioned and clamped as desired or can be taken out of the way if not needed. Many millers have round overarms, some have two. Braces are available to tie the arbor supports and overarm to the knee for added rigidity, as is done in Fig. 12-2.

The saddle, table, and knee may each be moved and positioned by hand by turning accurate screws with micrometer dials graduated to show 0.001 in. increments of movement. Each unit may also be moved by power at feed or rapid traverse rates. Many modern machines have controls at two stations, front and side, as in Fig. 12-1. Trip dogs can be set to disengage the feeds at desired limits.

An electric drive motor in a compartment in the column drives a countershaft by means of belts or chains. A clutch and brake on the countershaft are engaged by the long starting lever on top of the

column. Power is taken from the countershaft through a series of change gears in the column to drive the spindle. Power is also taken from the countershaft through another set of change gears and transmitted by shafts to feed the table, saddle, and knee.

Knee and column milling machines are provided with mechanical, hydraulic, or electric devices for changing speeds and feeds quickly. When a lever on the front of the saddle or side of the column of the miller of Fig. 12-1 is pushed, a hydraulic mechanism shifts the speed gears through successive positions. A dial on the side of the column indexes to show the changes. When the desired speed is shown by the dial, the lever is released. The feeds are changed in a similar manner.

The spindle of a milling machine is hollow with an accurate taper in the hole at the nose end. Older milling machines had Brown and Sharpe or other sticking tapers, but present-day machines have National Machine Tool Builders Standard Tapers, commonly called Milling Machine Standard Tapers, of 3½ in. per foot in sizes numbered 10, 20, 30, 40, 50, and 60. The higher the number, the larger the diameter and length of the taper. The nose of the spindle has an accurate outside pilot diameter, a true face, driving slots and keys, and bolt holes. Large face mills and some other cutters are bolted directly to the nose of the spindle, but most cutters are mounted on arbors or in adapters that have tapered shanks to fit the tapered hole in the spindle. They are held in the taper by a long draw bolt from the back end of the spindle. The outer ends of arbors are supported in the arbor support bushings.

The table has lengthwise T slots for bolts and keys, for fastening and positioning fixtures and clamps. The table, saddle, and knee move on well-proportioned ways and are gibbed snugly to move in accurate paths. Each unit can be clamped in place when it is to remain still.

The *universal* knee and column milling machine is like the plain type except that the table can be swiveled in a horizontal plane. This is especially necessary for milling a helix, as shown in Fig. 12-2, because a disk-type cutter must be set in line with the helix. The saddle is made in two parts. The upper part swivels on the lower and carries the table with it. Normally the table can only be turned about 45° before it bumps the column, and that limits the helix angles that can be cut without extra equipment. The leadscrew is

arranged so that a geared drive can be taken from it to a dividing head on the table.

An *omniversal* milling machine illustrated in Fig. 12-3 is designed particularly for toolroom and experimental work. In addition to all the adjustments common to the universal milling machine, the knee of the omniversal machine can be swiveled to tilt the table and can be fed horizontally. The machine has a spindle in the column

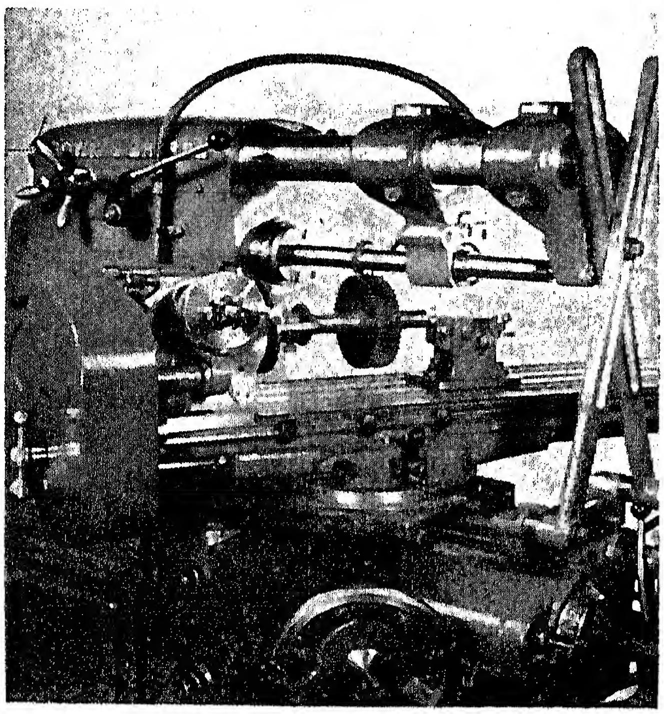


Fig. 12-2. A No. 2 universal knee and column type milling machine with a dividing head and helical milling attachment set up for cutting a helical gear. (Courtesy Brown and Sharpe Mfg. Co.)

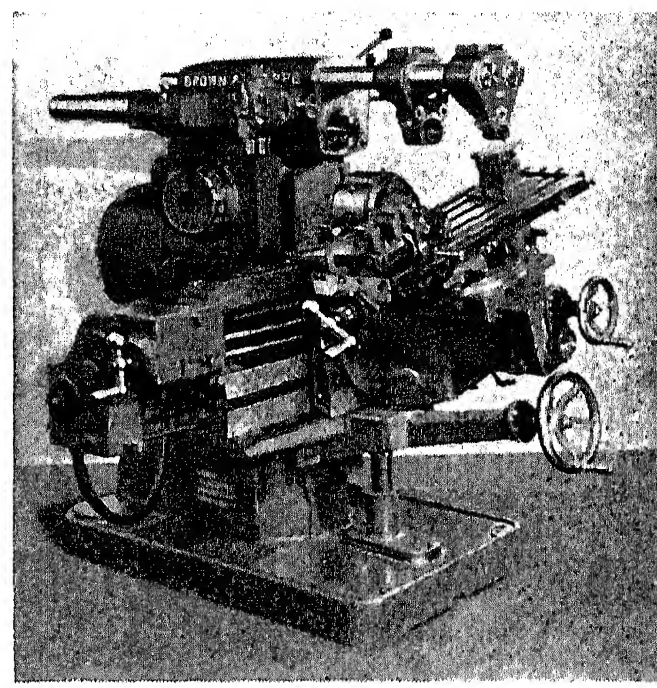


Fig. 12-3. A No. 0 omniversal milling machine. (Courtesy Brown and Sharpe Mfg. Co.)

and an auxiliary spindle head on the side of the column that can be adjusted in or out or inclined at any angle.

A *vertical* knee and column milling machine, as shown in Fig. 12-4, has a vertical head and spindle. On some models the head is fixed on the column, but on others, like the one shown, the head can be moved up or down. That machine is equipped with a turret stop and indicator to position the head accurately. On some vertical millers the head can be swiveled around a horizontal axis in a plane parallel to the movement of the table.

Sizes of knee and column milling machines. The sizes of knee and column milling machines are designated by numbers that in-

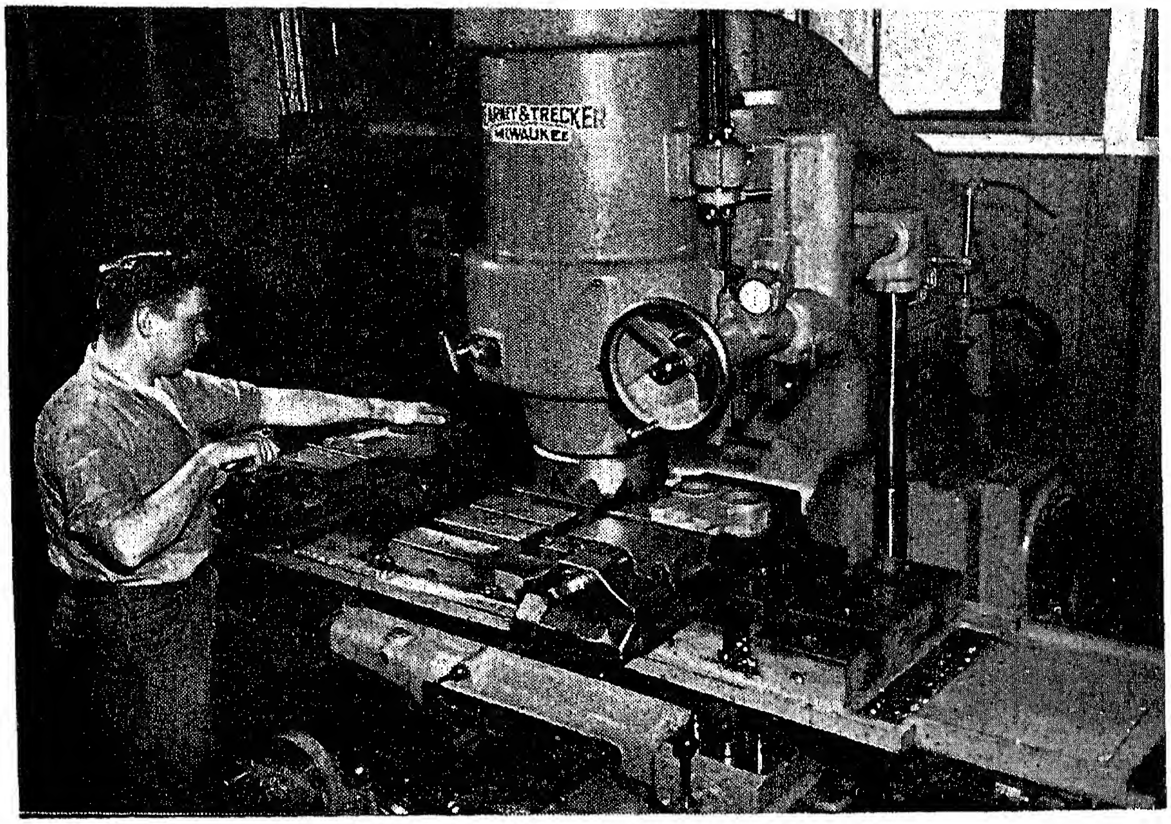


Fig. 12-4. A model CSM vertical milling machine engaged in face milling arbor supports. (Courtesy Kearney and Trecker Corp.)

dicate longitudinal table travel. The usual numbers for sizes and corresponding dimensions are:

Number	1	2	3	4	5	6
Table travel, inches	22	28	34	42	50	60

The actual sizes of the tables vary considerably among different makes and models of machines of the same nominal size.

Machines of each nominal size are built in light-, medium-, and heavy-duty models. The capacity of a machine is often designated by a letter or letters. For instance, one manufacturer offers a No. 2 knee and column milling machine with a 7½ hp motor, a No. 2 MI with a 5 hp motor, and a No. 2 ML with a 3 hp motor. Another makes a machine with a No. 2 table travel but with a 20 hp motor and corresponding ruggedness to utilize fully cemented carbide cutters.

Speeds and feeds provided vary with different models, makes, and sizes of milling machines. A typical No. 2 plain or universal miller has 16 speeds from 35 to 1400 rpm and 16 feeds from ½ to 20 in. per minute.

Because of their versatility and range, knee and column milling machines lack some of the qualities needed for high production. A certain amount of looseness in the joints between the major units and overhang are unavoidable. A tendency to chatter is found under heavy cuts. A number of separate controls are needed to govern all the available movements and require individual attention from the operator.

Production milling machines. Production or *manufacturing milling machines* are designed to remove metal rapidly and to require a minimum of attention from the operator. They are not as flexible in application as knee and column millers. For instance, adjustments are provided in three coordinate directions, but feeds are often available in only one direction and seldom in more than two directions. Flexibility is not so important because production millers are intended for long runs where setup changes are not frequent and special fixtures and cutters are often justified to adapt the machines to specific jobs.

Production milling machines are made in several styles. Some small models have knees, but the most common kind has a solid bed that forms a base for the major units and on which the table rides. They are sometimes called *bed-type milling machines.* As many of the units as possible are securely tied together. The construction is rigid to enable the machines to take heavy cuts. The operator is aided because the controls are simple and the action often automatic.

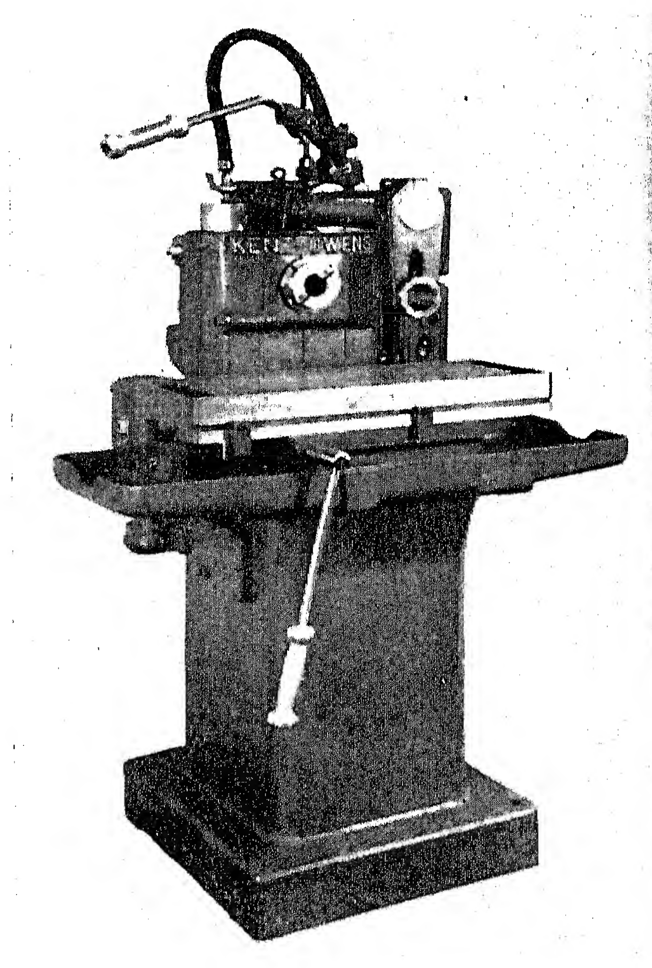


Fig. 12-5. A No. 1-M hand milling machine. (Courtesy Kent-Owens Machine Co.)

Hand-operated millers are simple and usually small. Many are bench mounted. They are suited for short light cuts, like slotting

the heads of screws. The table of the hand milling machine of Fig. 12-5 is moved by the feed lever that may be placed at the front as shown or at the rear. A rack on the table is engaged by a pinion on the lever shaft. Adjustable dogs are set to limit table travel. The cutter spindle is carried in a quill in the head and can be adjusted over a distance of 2½ in. in or out and locked in position. The head or

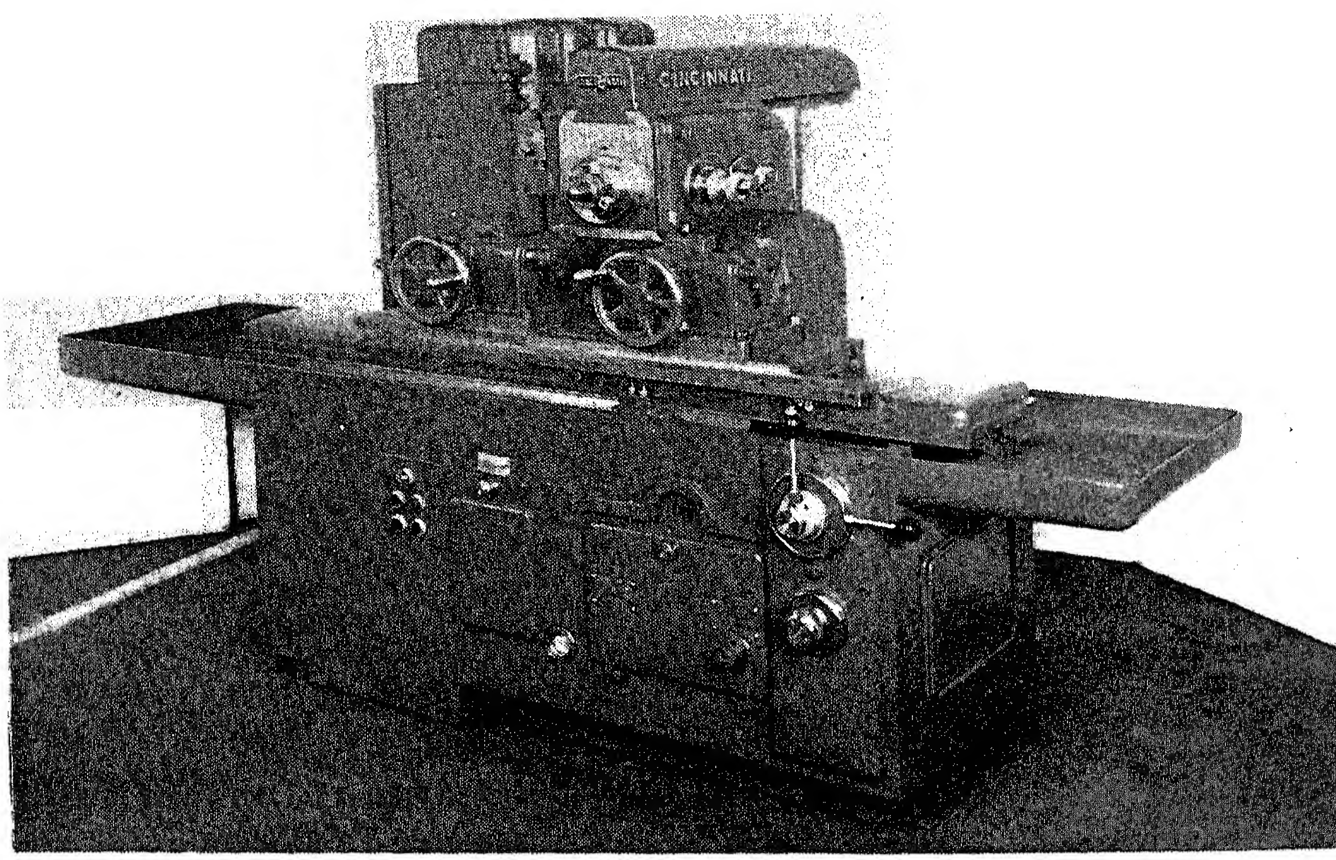


Fig. 12-6. A 2-24 plain automatic milling machine. (Courtesy Cincinnati Milling Machine Co.)

spindle carrier is mounted between two vertical posts. It can be raised or lowered by a counterbalanced lever on top to feed the cutter into the work or can be clamped to hold the cutter at a fixed height. The spindle is driven by a 1 hp motor through pick-off change gears. Speeds from 150 to 2000 rpm are available with an 1800 rpm motor.

The medium-sized production milling machine of Fig. 12-6 has a double set of fixtures on the table for a specific job. The table has a maximum movement of 24 in. longitudinally on the bed. Dogs on the front of the table are set to trip plungers to direct the table through

each cycle of the operation. The table may be fed in either or both directions. In a typical cycle, the table carries the work rapidly to the cutter, proceeds at slow feed, reverses at the end of the cut, and withdraws. The spindle is started and stopped at the beginning and end of each cut. Once the machine has been set up, all the operator has to do to control it is to start each cycle by pushing a lever on the front of the bed.

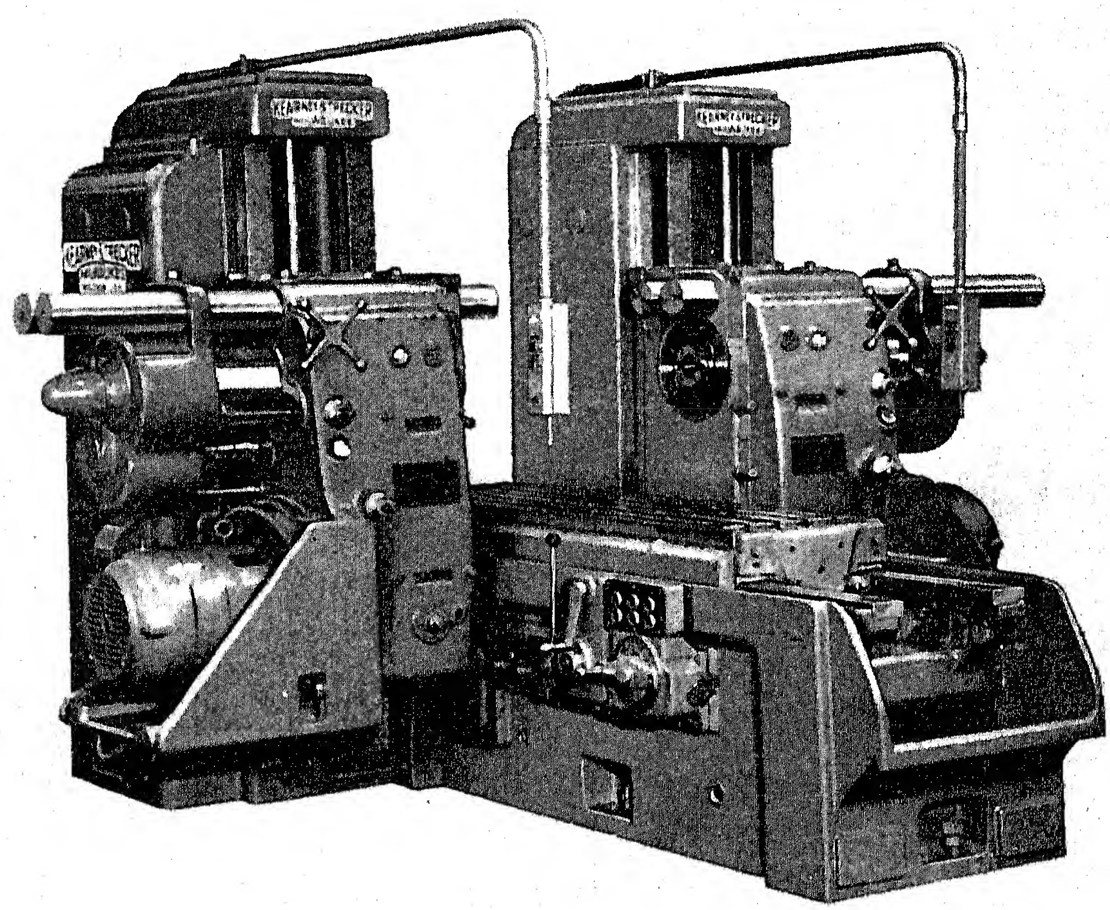


Fig. 12-7. A model CSM duplex bed type production milling machine. (Courtesy Kearney and Trecker Corp.)

The spindle is carried in a quill in the spindle carrier and is adjustable crosswise over the table. The spindle carrier is adjustable vertically on a headstock or column attached to the rear of the bed. After the spindle has been positioned for straight milling, the quill and spindle carrier are clamped in position. Machines of this kind are built with spindle carriers having controlled rise and fall movements. The cutter is raised and lowered at positions determined by trip dogs placed along the front of the table. By this means the

cutter can be fed down into the work or can be made to jump over obstructions as desired.

Some manufacturing millers have hydraulic table drives. The table of the machine of Fig. 12-6 is mechanically driven but hydraulically controlled. Both speeds and feeds are changed by means of pick-off gears.

The spindle carrier has an overarm for arbor supports when needed. It may be tied at its outer end to the bed by a brace for added rigidity. The bed contains a coolant reservoir and chip compartment.

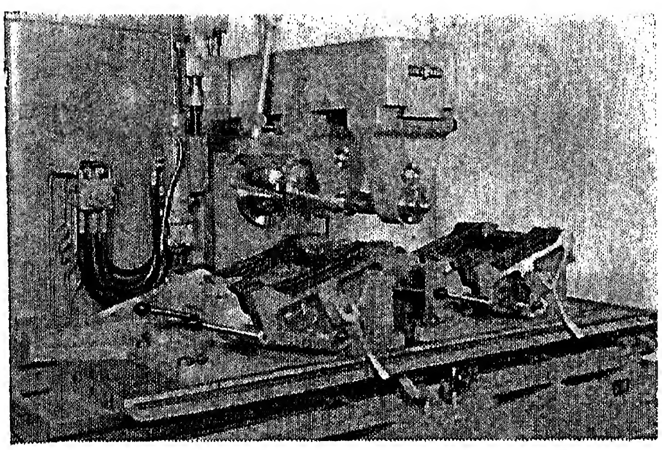


Fig. 12-8. A No. 5-60 plain tracer controlled Hydromatic production type milling machine. (Courtesy Cincinnati Milling Machine Co.)

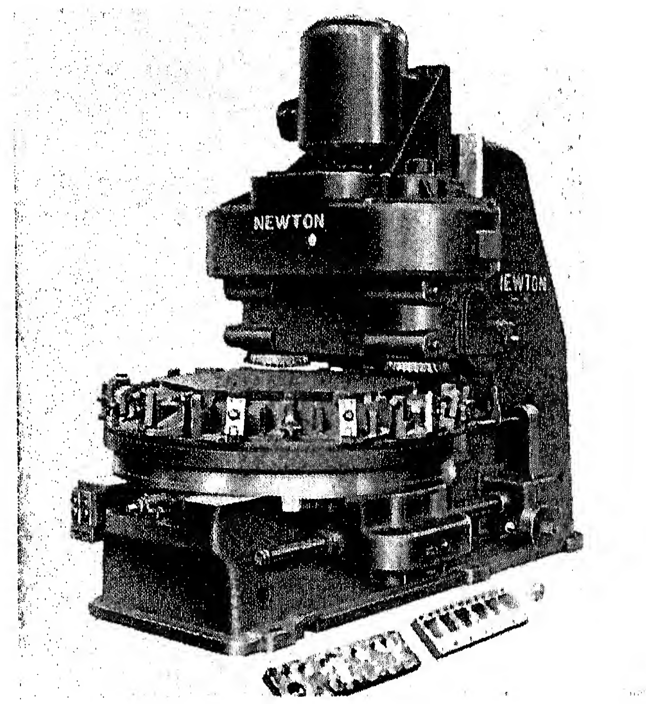


Fig. 12-9. A 76 in. vertical rotary miller. (Courtesy Consolidated Machine Tool Corp.)

The production miller of Fig. 12-6 is called a *plain* machine because it has one spindle carrier. A machine with two opposed spindle carriers is called a *duplex,* as is the one in Fig. 12-7. The spindles can be adjusted independently for cuts on two sides of workpieces fed between them. The miller of Fig. 12-7 is a heavy and high powered machine designed to get the most from cemented carbide cutters.

Tracer-controlled production milling machines are made to mill irregular surfaces in moderate to large quantities. A typical job is shown in Fig. 12-8. A template is bolted to the rear of the table alongside the work-holding fixtures. A roller on a tracer control valve on the side of the spindle carrier bears on the template. As the table moves, the spindle carrier is raised and lowered hydraulically

under the guidance of the tracer control valve. The cutters act in a path along the work corresponding to the profile of the template.

The milling machines so far described have had reciprocating tables. *Continuous millers* have been developed with rotary tables that feed the work continuously for high production. One kind is the *vertical rotary miller* shown in Fig. 12-9 for rough and finish milling the tops and bottoms of cylinder heads. The fixtures around the table are unloaded and loaded as they pass the operator at the front of the machine. A 40 hp motor drives both roughing and finishing cutters whose speeds are independently adjustable by means of pick-off gears. The table is on a saddle that may be moved in or out to alter the cut circle. The table feed may be continuous or may be made intermittent, to jump the gaps between pieces at a rapid rate.

Another type of continuous miller is the *drum type*. Its table revolves around a horizontal axis between two sets of opposed cutter heads.

Planer-type milling machines. Planer-type milling machines like the one in Fig. 12-10 are made to handle very large workpieces but

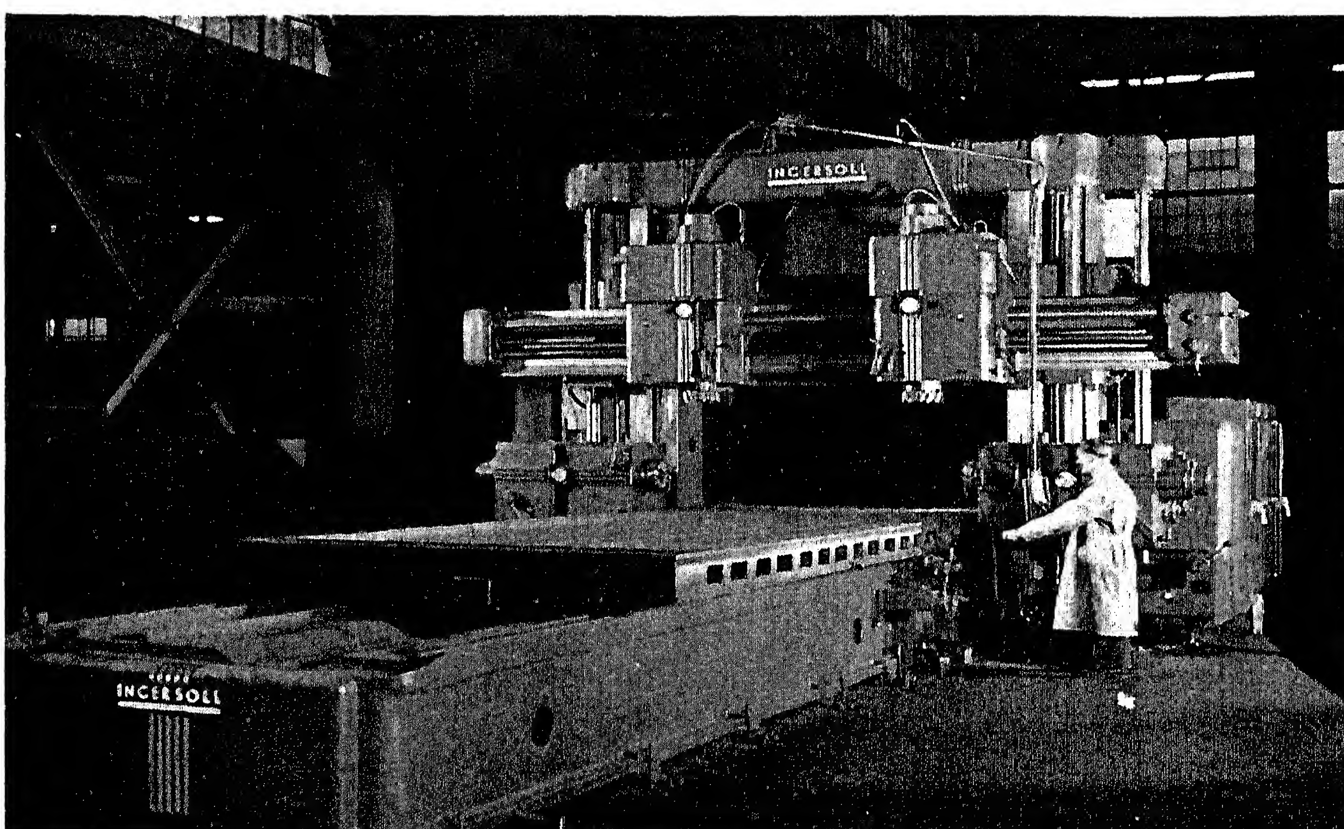


Fig. 12-10. A 96 in. by 60 in. by 20 ft planer type adjustable rail milling machine. (Courtesy The Ingersoll Milling Machine Co.)

have a flexibility approaching that of knee and column machines. The adjustable rail machine illustrated resembles a planer, but its table travels slower, and it has spindle carriers for rotating cutters instead of planer tool heads.

Special purpose milling machines. Some milling machines are made for special kinds of work. Important machines of that type

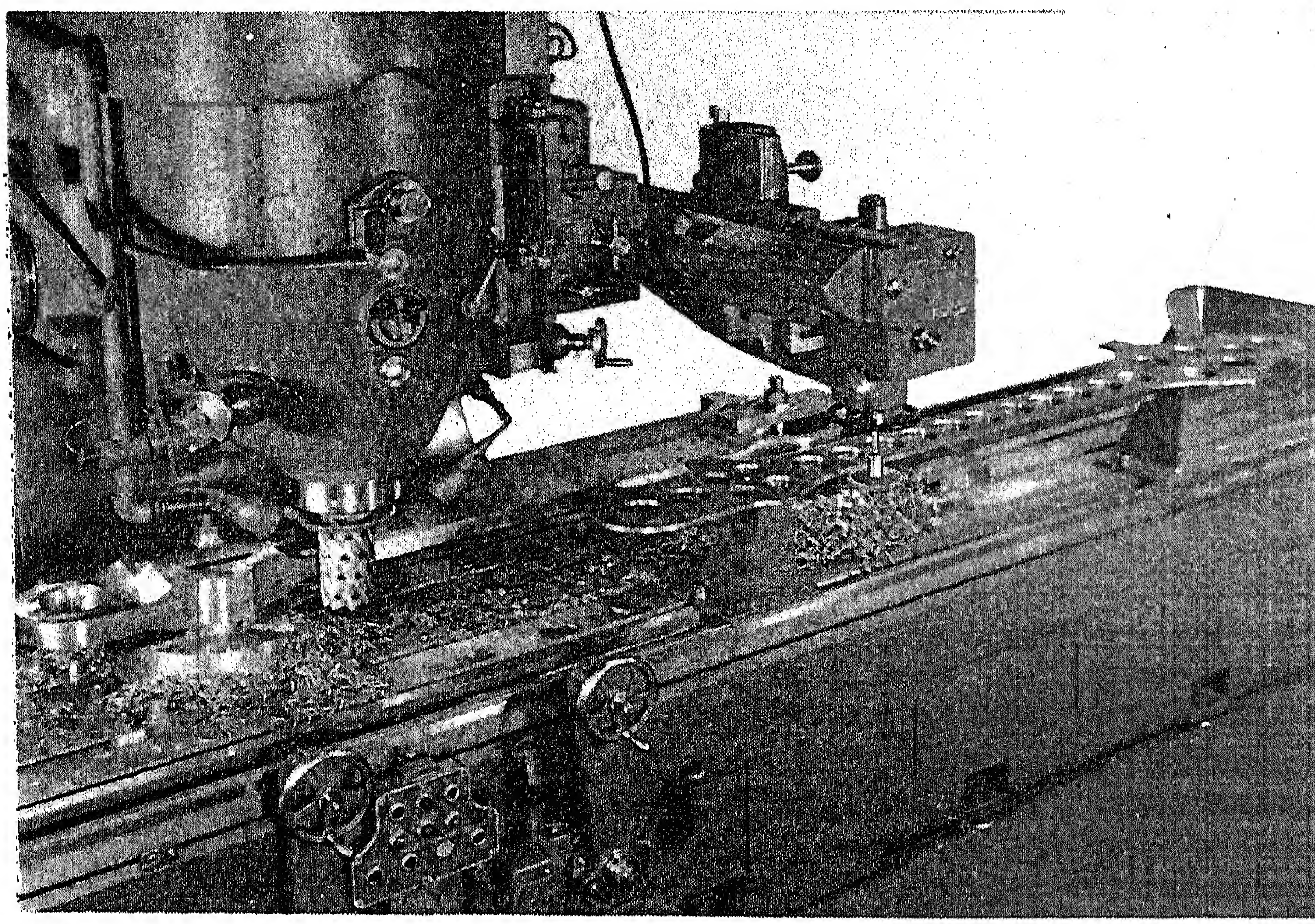


Fig. 12-11. A 36 in. vertical Hydrotel milling machine with 360° automatic profiling unit. (Courtesy The Cincinnati Milling Machine Co.)

are duplicating machines, cam millers, and planetary millers. Other milling machines are built for specific single jobs.

Profilers are milling machines capable of duplicating external or internal profiles from templates in two dimensions. Some are hand operated, like the machine of Fig. 12-12 which often is used for profiling. A hydraulic milling machine arranged for profiling locomotive side rods automatically is shown in Fig. 12-11. The table moves longitudinally, and the cutter head moves across the table under the direction of the tracer finger riding the template profile.

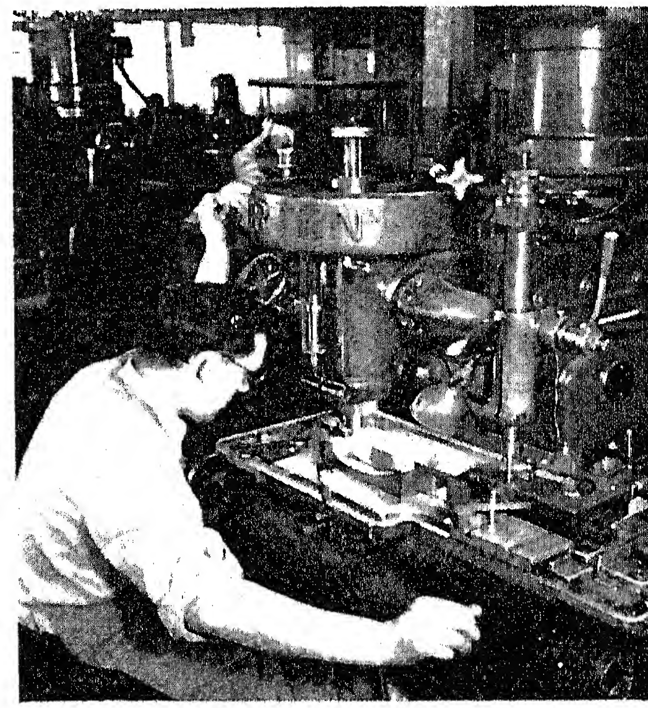


Fig. 12-12. A form being duplicated by hand manipulation. (Courtesy Geo. Gorton Machine Co.)

A *duplicator* reproduces forms in three dimensions. Duplicators are commonly used to make forging dies, form dies, steel molds, etc., and often are called *die sinking machines*. A hand-operated machine on which a form is being duplicated is shown in Fig. 12-12. The tracer finger is carried in a bracket attached to the spindle head and is moved up and down in unison with the cutter spindle quill. The operator is controlling this movement with his left hand. His right hand grasps a lever by which he moves the table in a horizontal plane. The master form and workpiece are clamped to the table. The form the cutter produces in the work is determined by the paths the tracer finger traces over the master form.

A setup for automatically milling a mold for a plastic serving tray is shown in Fig. 12-13. The tracer finger rides on the master form on the right and hydraulically controls the depth of cut in the workpiece on the left. Both master and workpiece are revolved on the power-driven duplex rotary tables while a fine cross-feed also operates.

Automatic profilers and duplicators are made with multiple spindles. They are capable of reproducing the same form on several workpieces at one time. Some machines have horizontal rather than vertical spindles. Electrical controls are used on some makes.

Cam millers produce disk cams. The profile of the cam is cut on a slowly revolving workpiece by an end mill positioned by a master cam revolving in unison with the workpiece.

Planetary millers do internal and external circular form and thread milling. Inside and outside circular forms, an internal form and an external thread, or vice versa, may be milled at the same time, together with a face if desired. Even internal and external threads of different pitches and hands may be milled in the same operation.

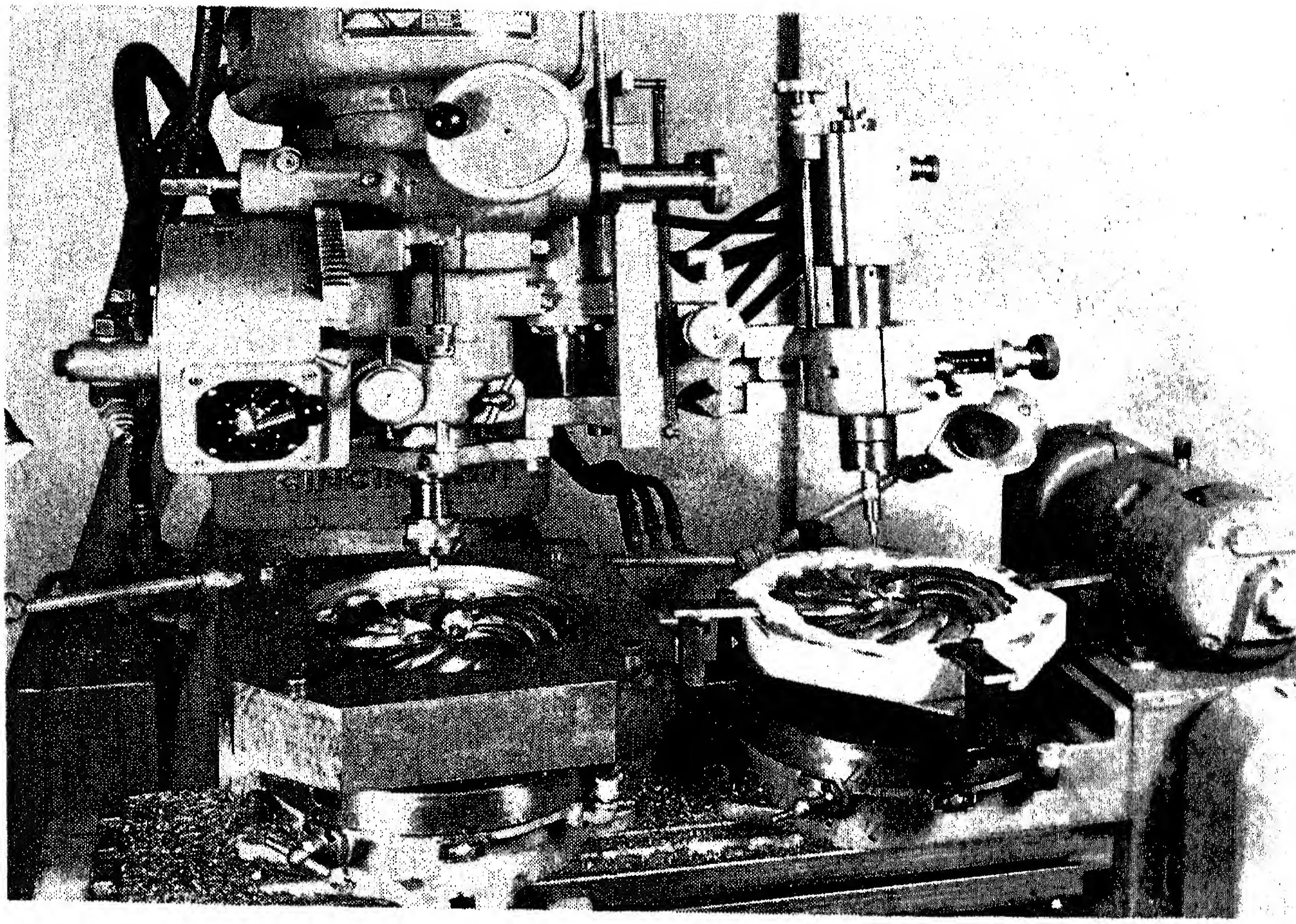


Fig. 12-13. An 8 in. by 18 in. tool and die milling machine equipped with a duplex rotary power driven table attachment. (Courtesy The Cincinnati Milling Machine Co.)

Planetary millers are used for all sizes and shapes of workpieces but are particularly convenient for cumbersome pieces, because the work does not need to move.

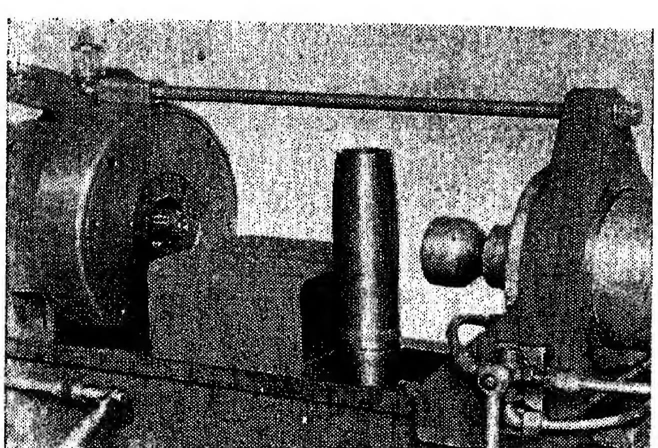
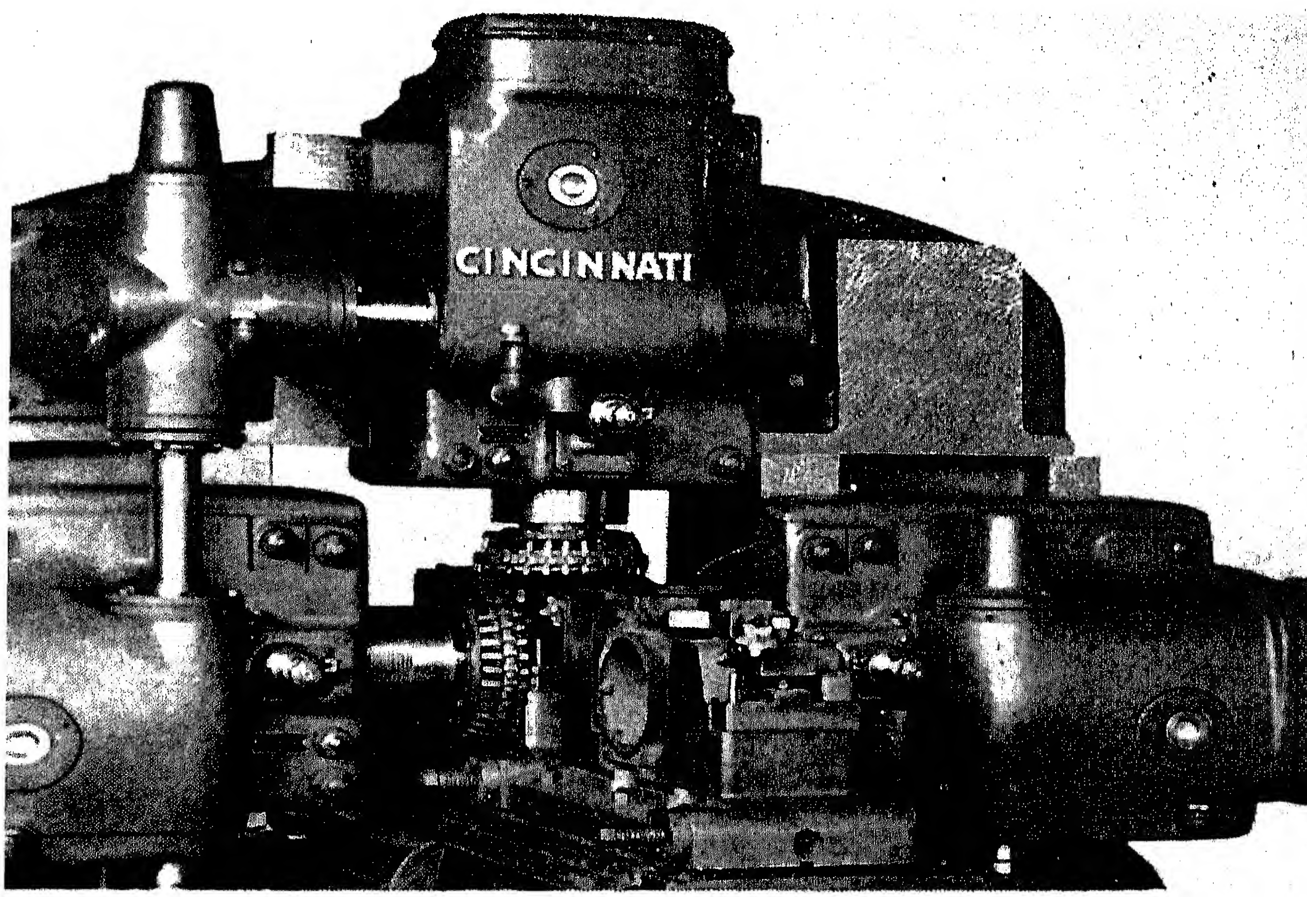


Fig. 12-14. A planetary miller arranged for cutting internal threads in a shell. (Courtesy The Hall Planetary Co.)

A planetary miller is set up for cutting internal threads in a shell in Fig. 12-14. The workpiece is pressed into a locating ring around the cutter by an air-operated tailstock. The cutter revolves around its own axis at cutting speed. At the beginning of the cycle, the cutter is moved off center to bring it to the proper depth in the work. Then the cutter is fed in a circular path around the axis of the work-

inally is retracted at the end of the cut. When threads the cutter is advanced an amount equal to its lead in one of the circular path. Some planetary millers have a head l so work can be done at both ends of a workpiece at the

illing machines are often built to do specific jobs quickly



5. A special milling machine constructed mostly of standard units he top as well as the sides of cast iron meter bodies in one operation. he Cincinnati Milling Machine Co.)

ntly when production is large enough to justify the initial ost every special milling machine is different from every ome respect.

milling machines constructed as much as possible from units are less expensive than entirely special machines. nits also can be salvaged if the machine becomes obsolete. milling machine built from a standard duplex production chine by the addition of a special bridge-type fixed height third standard spindle carrier is shown in Fig. 12-15.

Milling Cutters and Drivers

Features of cutters. Milling cutters of many types are made in many sizes. They differ according to the following principal features:

1. The size of a milling cutter is usually designated by its diameter. The length or width and the number of teeth also are commonly included in the description of a milling cutter.
2. Milling cutters may be of the *solid, tipped,* or *inserted tooth* types. A solid cutter has teeth integral with its body. Other cutters, particularly large ones, have teeth of expensive material inserted and locked in slots or holes in a soft steel or cast iron body. The inserted teeth can easily be replaced when worn or broken.
3. A milling cutter may have straight or helical teeth. Straight

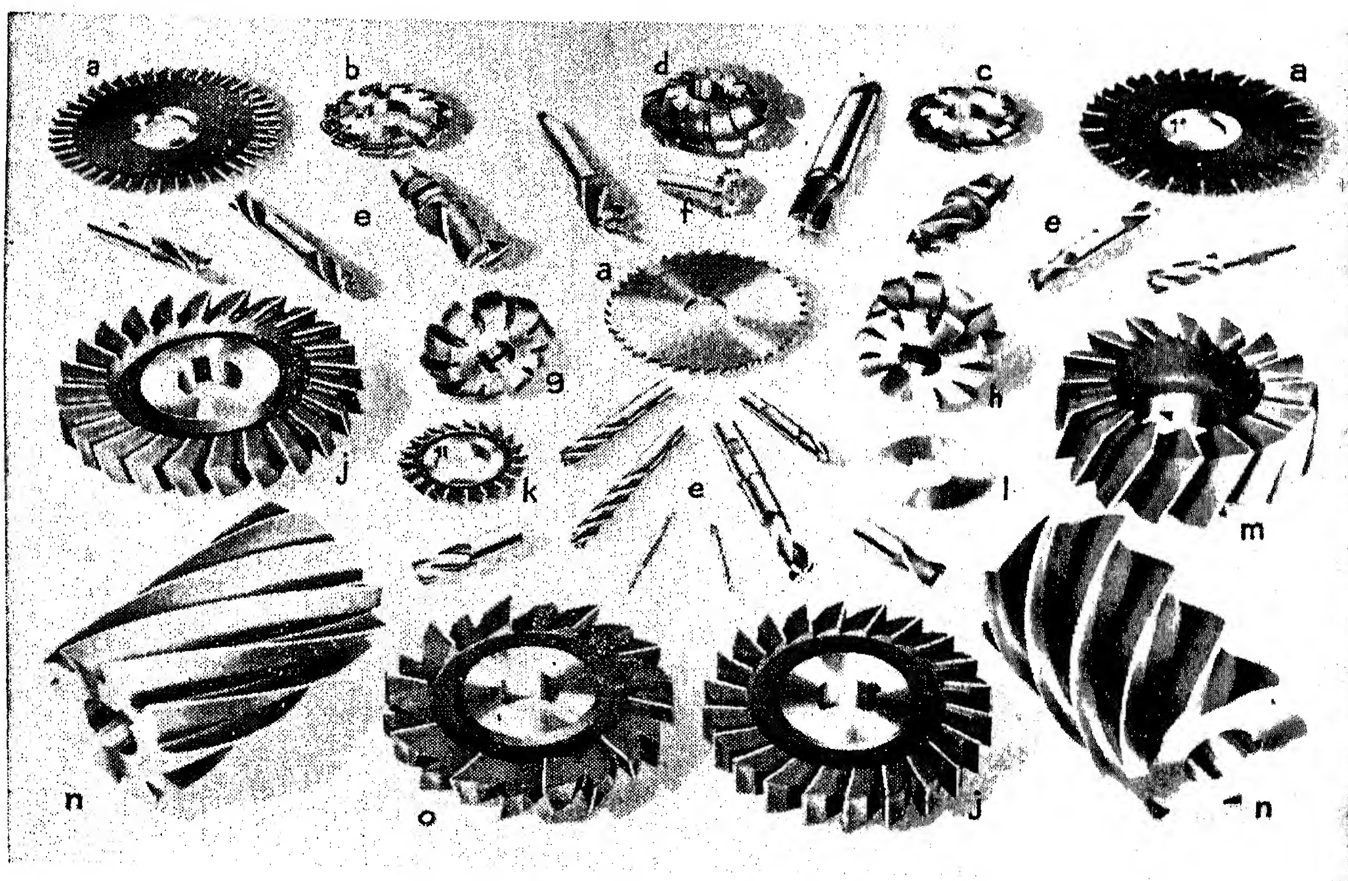


Fig. 12-16. Principal types of standard milling cutters. (Courtesy Brown and Sharpe Mfg. Co.) a. Metal slitting saws. b. Stocking cutter for involute spur gears. c. Involute spur gear cutter. d. Corner rounding cutter. e. End milling cutters. f. Woodruff keyseat cutters. g. Convex cutter. h. Concave cutter. j. Side milling cutters. k. Angular cutter. l. Screw slotting cutter. m. Shell end mill. n. Plain milling cutters. o. Staggered tooth milling cutter.

teeth are easier to sharpen but helical teeth cut more smoothly and produce better finishes because they enter and leave the work gradually. They may be right- or left-hand like a screw.

4. Hand refers to the direction a milling cutter must rotate in order to cut rather than rub. A right-hand cutter must rotate counterclockwise, a left-hand cutter clockwise. The hand is determined by looking at the front of the cutter when it is mounted on the spindle. The hand of rotation and the hand of the helix of a milling cutter are not always in the same direction. One cutter with teeth on a right-hand helix may have a right-hand cut, another a left-hand cut. Also a left-hand helix cutter may have either direction of cut.
5. Some milling cutters have a center hole for mounting on an arbor. Some have tapered or straight shanks. Others have a recessed back to fit the spindle nose. A few have threaded holes to be screwed on adaptors or arbors.
6. Some milling cutters are standard, others special. Standards for types and sizes of milling cutters have been defined by the American Standards Association and adopted by manufacturers. The principal standard types are shown in Fig. 12-16. Some have names that describe the work they do.

Types of milling cutters. *Plain milling cutters* cut only on their peripheries and are used to mill flat surfaces. The operation is called *slab milling* if the cut is wide. The teeth may be straight or helical. *Light-duty plain milling cutters* have closely spaced teeth with small chip spaces and take light cuts. Cutters less than ¾ in. wide usually have straight teeth. On wider ones, the teeth usually have less than a 25° helix angle. *Heavy-duty plain milling cutters* are often called *coarse tooth milling cutters* because they have widely spaced teeth with ample chip spaces. Cutter width is usually greater than diameter, and helix angles range from 25° to 45°. Cutters having helix angles over 45° are called *helical plain milling cutters.*

Large plain milling cutters with inserted teeth are called *slab mills.* Notches often are put on the blades and serve as chip breakers. Interlocking slab mills, one with a right-hand and the other with a left-hand helix, are desirable for wide cuts to break up the chips and neutralize end thrust.

Side milling cutters have cutting edges on one or both sides as well as on their peripheries. They may be used for cutting keyways, slots, and the sides of workpieces. *Plain side milling cutters* have cutting edges on both sides. A *half side milling cutter* has cutting edges on one side in addition to the periphery. The teeth may be straight or helical. These cutters are efficient for heavy side or straddle milling.

Staggered-tooth milling cutters have cutting edges alternately on one side and then on the other side of their teeth. Alternate teeth also have opposite helix angles. The peripheral cutting edge of each tooth runs from its side cutting edge only part way across the cutter. These cutters cut cleanly, provide good chip clearance, and take high speeds and feeds. They perform well in deep slots but are limited to narrow widths. The effective width of the cutter is changed when the side cutting edges are reground.

Two side or half side milling cutters milling both sides of a workpiece at the same time are said to be *straddle milling*, as is done in Fig. 13-7. Where several cutters are mounted on the same arbor, particularly to mill several surfaces on one workpiece, the operation is known as *gang milling*.

A set of *interlocking slotting cutters* contains two side or staggered tooth cutters placed side by side with overlapping teeth. Shims are inserted between the hubs of the cutters to adjust their over-all width and compensate for wear and sharpening loss. A pair of interlocking cutters is shown in Fig. 13-6.

Metal slitting saws are thin plain or side milling cutters from $\frac{1}{32}$ to $\frac{3}{16}$ in. wide. Their sides are often dished to reduce rubbing. Some saws have staggered teeth for heavy cuts. Typical operations for saws are cutting off, slotting screw heads, and milling deep narrow slots. Saws are fragile and should be run at about $\frac{1}{4}$ to $\frac{1}{8}$ of the feed per tooth that wider plain milling cutters will stand.

Angle milling cutters or *angular cutters* are shaped like truncated cones and are used for milling dovetails, V notches, serrations, reamer teeth, etc. A *single angle cutter* has cutting edges on the sides and base of a frustrum of a single cone with a base angle of 45° or 60°. The cutting edges of a *double angle cutter* are elements of two conical frustrums having a common base, as depicted in Fig. 12-16. The teeth are V shaped with conventional angles of 45°, 60°, and 90°.

End milling cutters or *end mills* are solid cutters with cutting edges on the periphery and over most or all of one end. Their teeth may be straight or helical. The smaller sizes have integral shanks, some straight and some tapered. Several kinds of end mills are shown in Fig. 12-16. They are used for light cuts in facing, profiling, boring, and slotting operations. Applications are shown in Figs. 12-6, 12-12, and 12-13.

Two lip end mills or *slotting mills* have two flutes. Their end cutting edges extend to the center, and they are the only end mills capable of drilling into solid metal. The large sizes of end mills commonly do not have integral shanks. They are called *shell end mills* and have a center hole for mounting on stub arbors and a keyway on one end. Some are made with inserted teeth.

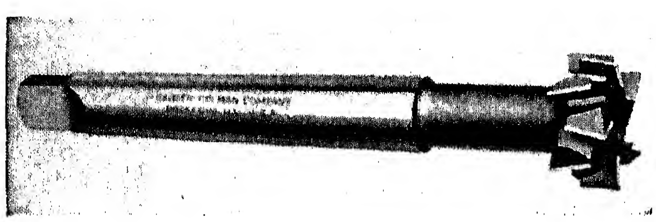


Fig. 12-17. A T slot cutter. (Courtesy Barber Colman Co.)

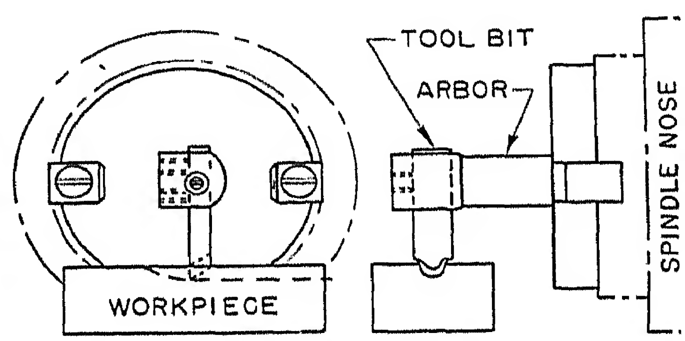


Fig. 12-18. A sketch of a flycutter.

T-slot milling cutters are used primarily for the specific operation of undercutting T slots. A T-slot cutter is shown in Fig. 12-17. Large sizes may have separate shanks. The center of a T slot is opened first with an end mill. Then the T head is cut with a light feed because the T-slot cutter head is almost buried in metal.

Woodruff key milling cutters are used to cut keyseats. The sizes up to 2 in. diameter have integral shanks and straight teeth with peripheral cutting edges only. The sides of the teeth are slightly relieved for clearance. Larger sizes are staggered tooth milling cutters with a hole for arbor mounting. They are made in sizes corresponding to Woodruff key slot sizes.

A *fly cutter* has a single tooth. The bit may be held in an arbor, as shown in Fig. 12-18, or in a heavy disk that acts like a flywheel to store energy for the cut. A single point tool is relatively easy to form and can be made to do accurate work. Fly cutters are used chiefly for form milling where only a few pieces are to be made and the cost of a special multiple tooth cutter is not justified.

Face milling cutters or *face mills* are used for facing wide surfaces. They generally are over 6 in. in diameter and are bolted directly to the spindle nose. Applications of face mills are shown in Figs. 12-4, 12-9, 12-10, and 12-15. Cutting material is brazed directly to recesses in the bodies of some face mills, but mostly teeth of high speed steel, cast nonferrous alloy, or cemented carbide are inserted and locked in a body of soft steel or cast iron. The teeth or blades may differ in shape in one cutter body so that some act as roughers and others as finishers. Face mills may be designated as heavy-duty or light-duty, according to the strength of the body. Heavy-duty face mills have fewer teeth and are suited for roughing. Light-duty face mills are used mostly for finishing.

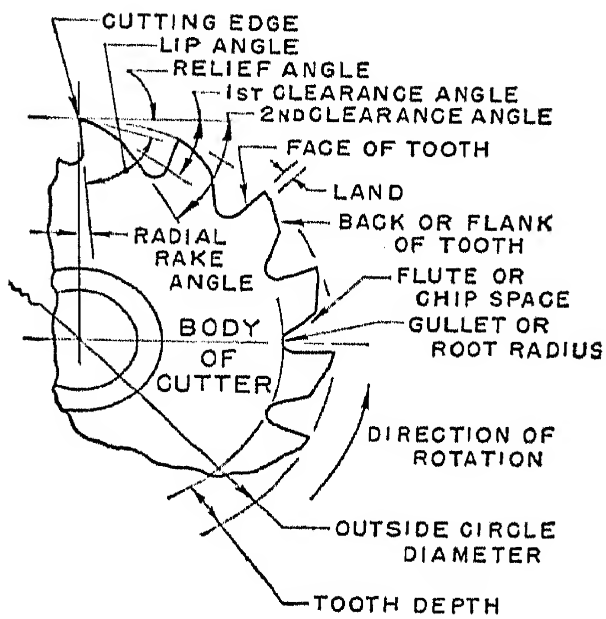


Fig. 12-19. The elements and angles of the teeth of a plain milling cutter.

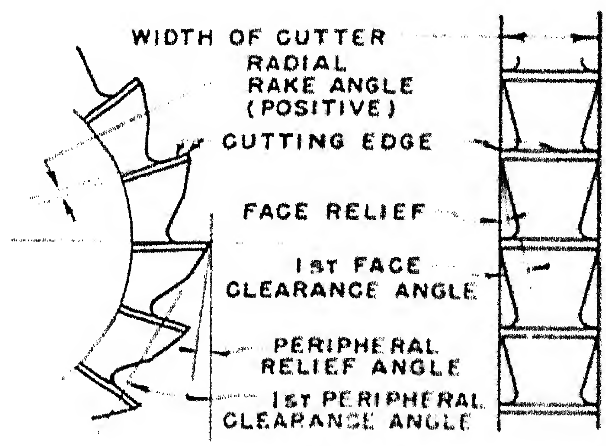


Fig. 12-20. Angles of a side milling cutter.

A *formed milling cutter* usually has curved tooth edges and cuts a surface with a specific profile or contour. Most formed milling cutters are special, but some are made in standard sizes and shapes. The latter include cutters for gear, spline, and chain sprocket teeth, radii, and threads. Two common types of formed cutters are shaped or formed profile and form or cam relieved cutters.

A *shaped* or *formed profile cutter* is also called a *profile ground cutter* and has a cutting edge of the desired form on each tooth with a narrow land behind the edge. The land and edge are reground to

the desired form each time the cutter is sharpened, preferably to match a profile gage and master. That may not be difficult for simple profiles but is difficult for complex forms. These cutters may be solid or have inserted blades.

A *form* or *cam relieved cutter*, also called a *face ground cutter*, has a cutting edge of the desired form around each tooth and also the same profile in radial or parallel planes behind the cutting face. Each tooth recedes uniformly behind its edge for relief. The tooth form is produced by turning the cutter with a form tool that is fed in and retracted by a cam for each tooth. The cutter is sharpened by grinding the same amount from the faces of all the teeth. The faces may be radial or have a rake angle, but the amount of rake is kept the same when the cutter is sharpened. The teeth of some cutters have a helix angle of 5° to 10°. Form relieved cutters are comparatively easy to sharpen, and their teeth may be ground until they become too frail to withstand the cutting load.

Teeth of milling cutters. The teeth of a milling cutter have angles like other cutting tools. The names of the surfaces or *elements* and angles of a plain milling cutter are given in Fig. 12-19. The *land* behind the cutting edge is ground to sharpen the cutter. The land width varies from 1/64th in. for small diameter cutters to 1/16 in. for large cutters. The *relief angle* between the land and the tangent to the radius keeps the tooth from rubbing behind the edge. That angle in general is from 3° to 7° for iron, steel, and bronze and as high as 12° for brass, aluminum, and magnesium. The tooth is backed off behind the land with *1st and 2nd clearance angles*. Form relieved cutters do not have these definite clearance angles.

The *face* of a tooth is the surface between the fillet and the cutting edge on which the chip is formed. It corresponds to the top of a single point tool. The *radial rake angle* is the angle between the face of the tooth and a radial line through the cutting edge. A rake angle in the direction indicated in Fig. 12-19 is positive. In general, rake angles are from 10° to 15° for iron, steel, and bronze and from 20° to 35° for aluminum and magnesium. Some cutters have no rake.

The *flute* is the chip space between the back of one tooth and the face of the next. It provides room for the chip during a cut. Ample chip space is necessary for heavy cuts, especially if a continuous curled chip is formed from ductile material. The form of the chip

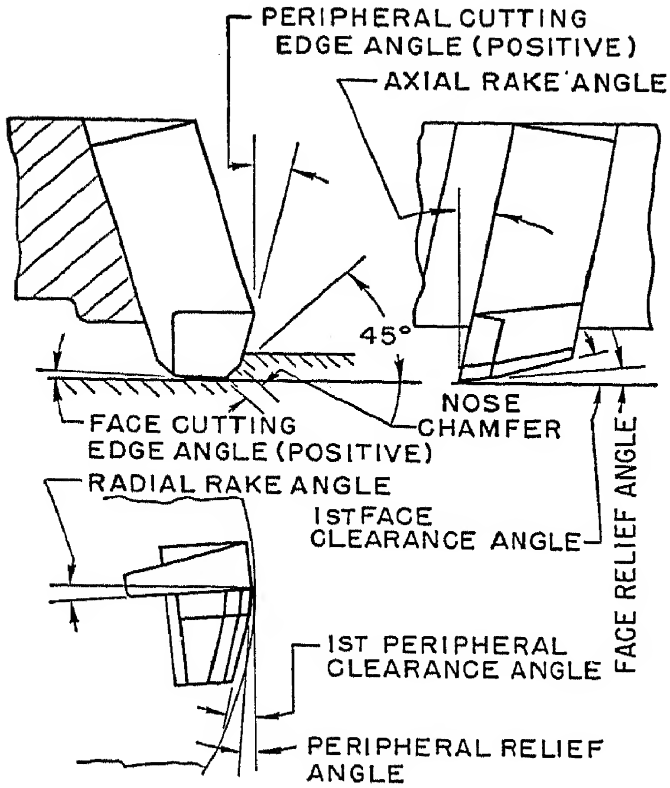


Fig. 12-21. Angles of a typical face mill.

space is also important. A large radius at the bottom of the chip space helps prevent clogging or wedging of chips.

Other kinds of cutters have surfaces and angles corresponding to those described for the plain milling cutter. A diagram of the teeth of a side milling cutter is shown in Fig. 12-20. One type of face mill is depicted in Fig. 12-21. The teeth are adjustable axially for face wear resulting from shallow cuts. They have axial and radial rake angles that determine the *true rake angle* in a plane perpendicular to the cutting edge. Negative rake angles are recommended for cemented carbide teeth to cut steel. They increase power consumption a little but help reduce the shock on the cutting edges.

Detailed recommendations for milling cutter tooth angles for various applications are given in reference books and handbooks Typical cutter grinders for sharpening milling cutters are described in Chapter 17.

Arbors, Collets, and Adapters

A variety of holders and drivers is needed to accommodate the many sizes and types of cutters to milling machines. These are known as milling machine arbors, collets, and adapters.

Milling machine arbors. A milling machine arbor has a tapered shank, a flange with keyways, and an accurate straight diameter on which cutters are mounted. A keyway runs along the straight portion. Three standard styles of arbors are shown in Fig. 12-22. The tapered end of an arbor has a threaded hole for a drawbar that holds the arbor in the spindle.

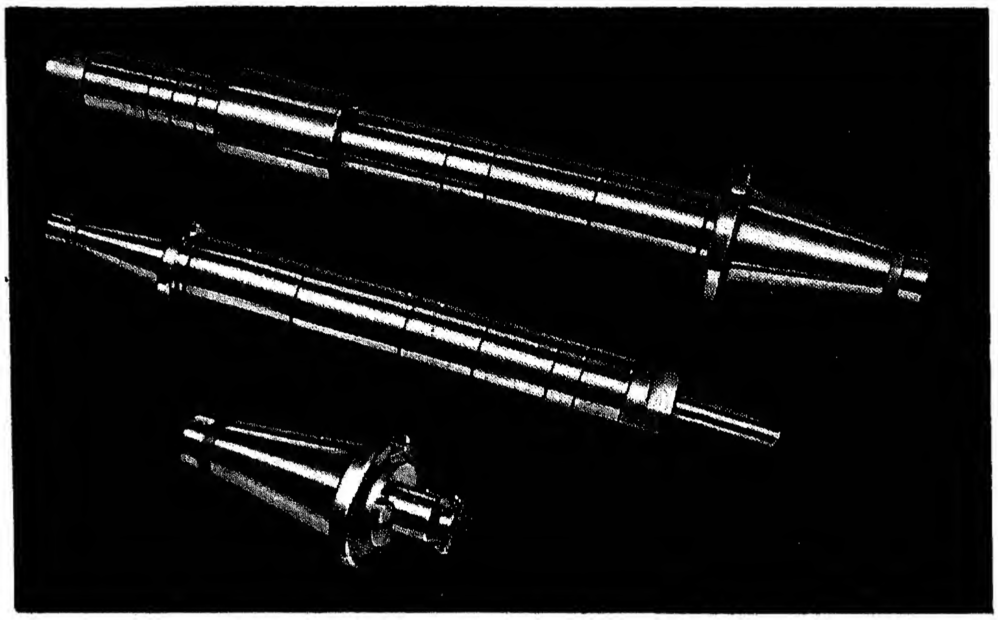


Fig. 12-22. The three styles of milling machine arbors. (Courtesy Kearney and Trecker Corp.)

Styles A and B arbors have spacing collars to position cutters. Both cutters and collars fit snugly on the arbor and are drawn together tightly by the nut on the end of the arbor. The ends of each spacing collar are ground parallel within close limits because on that depends the running truth of the arbor. If an arbor is to be kept accurate, its collars must be clean and free of burrs and nicks at all times. Several lengths of spacing collars are supplied with each arbor. They are arranged to position cutters as desired on the arbor. In addition, the spacing between cutters may be adjusted to a thousandth of an inch by shims or adjustable collars.

The style A arbor in the middle of Fig. 12-22 has a small diameter pilot on its outer end. That pilot runs in a style A arbor support like the one supporting the outer end of the arbor in Fig. 12-2. That provides the most clearance for a fixture, vise, or workpiece that must pass beneath the arbor support and is particularly helpful if the cutter diameter is small. Style A arbors may also carry bearing collars to fit style B arbor supports.

The style B arbor at the top of Fig. 12-22 does not have an end pilot but one or more bearing collars, larger in diameter than the spacing collars. A style B arbor with bearing collars rotating in two arbor supports is shown in Fig. 12-8. The bearing collars provide a large bearing area and can be placed close to or between cutters for maximum rigidity under heavy cuts.

The style C arbor at the bottom of Fig. 12-22 is for shell end and small face milling cutters. It is sometimes called a *shell end mill arbor*. The short straight portion fits into the hole in the cutter. Keys on the flange enter a keyway across the back of the cutter. A lock nut or bolt on the end clamps the cutter against the flange of the arbor.

Any arbor is designated by a series of numbers and letters that specify in order its taper size, straight diameter, style, length from flange to nut, and size of bearing. For example, a 51½B24-4 arbor has a No. 50 NMTBA tapered shank, takes cutters with 1½ in. diameter holes, is style B, has a length of 24 in. from flange to nut, and carries a No. 4 bearing.

Collets and adapters. Collets and adapters extend the range of taper sizes that may be put in a milling machine spindle. A collet or adapter has an outside taper and a concentric tapered hole.

An adapter usually has an outside milling machine standard taper and a smaller inside taper. For instance, several different adapters may have a No. 50 milling machine outside taper. One may have a No. 40 milling machine taper hole for small arbors. Another may have a Brown and Sharpe taper hole for the shanks of milling cutters. Still another may have a Morse taper hole for drills. An adapter may be held in a spindle by a draw bar or be bolted to the spindle nose. The end mill in Fig. 12-6 is held by an adapter.

Collets usually but not always have sticking tapers, such as Brown and Sharpe or Morse tapers, inside and outside.

A *quick change adapter* with *quick change collets* is convenient when a number of cutting tools is used in an operation because the tools may be changed in the spindle without screwing and unscrewing the draw bolt each time. Each tool needed, such as a drill, end mill, boring bar, or arbor, is held in a separate quick change collet. Each collet has a tapered shank that fits the hole in the adapter mounted on the spindle nose. Lugs on the collet flange are driven by slots in the adapter body. A slotted ring nut on the adapter is given a half turn to hold or release a collet.

A *spring chuck or collet holder* may be mounted on a milling machine spindle to hold wire, rods, and straight shank tools. A spring collet of the proper size is inserted and tightened by a cap nut on the end of the chuck.

A *centering plug* fits in the tapered hole of a milling machine

spindle and has a straight diameter to pilot cutters bolted to the spindle nose. A large face mill is easier to mount if a centering plug is used, and the plug also helps to keep the spindle hole clean.

Questions

1. Name the important units of a plain knee and column milling machine and describe the purpose of each.
2. How does a universal differ from a plain knee and column milling machine?
3. What feature does a vertical knee and column milling machine have?
4. What are the advantages and disadvantages of a knee and column milling machine?
5. How is a production milling machine designed to suit its purpose?
6. What is a duplex manufacturing milling machine?
7. Describe a continuous miller.
8. What is the difference between a profiler and a duplicator?
9. Describe a planetary miller. What can it do?
10. What advantages result from building special milling machines with standard units?
11. What are the principal features of milling cutters?
12. Name and describe the chief types of milling cutters.
13. Sketch teeth of a plain milling cutter and identify their elements.
14. Name and describe three standard style arbors.

References

American Standards Association, New York, N.Y.:
Milling Cutters, ASA B5.3–1950
Nomenclature for Milling Cutter Teeth ASA B5c1–1947
Spindle Noses and Arbors for Milling Machines ASA B5.18–1943

The Milling Machine and Its Attachments. Milwaukee, Wis.: Kearney and Trecker Corp., 1945.

A Treatise on Milling and Milling Machines. Cincinnati, Ohio: The Cincinnati Milling Machine Co., 1951.

Manufacturers' Catalogs:
The Brown and Sharpe Mfg. Co., Providence, R. I.
The Cincinnati Milling Machine Co., Cincinnati, Ohio.
The Consolidated Machine Tool Corp., Rochester, N.Y.
George Gorton Machine Co., Racine, Wis.
Hall Planetary Co., Philadelphia, Pa.
Ingersoll Milling Machine Co., Rockford, Ill.
Kearney and Trecker Corp., Milwaukee, Wis.
The Kempsmith Machine Co., Milwaukee, Wis.
Kent Owens Machine Co., Toledo, Ohio.

Chapter 13

MILLING ATTACHMENTS AND OPERATIONS

MANY STANDARD ATTACHMENTS ARE AVAILABLE to make milling machines easier and faster to operate and increase the variety of jobs that can be done. Attachments generally are added to general-purpose milling machines but sometimes are found on production machines to facilitate specific jobs. These attachments may be divided into two general classes. One class includes those for positioning and driving cutters. The other includes attachments for positioning and holding work.

Cutter Driving Attachments

Vertical milling attachment. A vertical milling or *swivel head attachment* adds to the versatility of a horizontal miller by transforming it when needed into a vertical miller. The vertical attachment of Fig. 13-1 is mounted on the face of the column and supported by the overarm. It is proportioned to take reasonably heavy cuts. Its spindle is driven by and at the same speed as the horizontal machine spindle. Some special vertical milling attachments have two or more spindles.

Universal spiral milling attachment. A universal spiral milling attachment typified in Fig. 13-2 can be swiveled in two planes through 360° on some models, less on others. The attachment is bolted to the machine column and is commonly supported by an arbor support. The attachment spindle is driven by the machine spindle, and both spindles are made to rotate at the same speed on most models.

A universal milling attachment is useful for setting cutters at compound angles for die, jig, or fixture milling. An important use is for milling helical gears, twist drills, and threads. The attachment even adds to the range of a universal milling machine by making possible the milling of helix angles greater than 45°, the normal limit obtained by swiveling the table alone.

High speed milling attachments. A high speed attachment is added to a milling machine to drive small cutters at high speeds for light work such as die sinking and keyway cutting. One type is

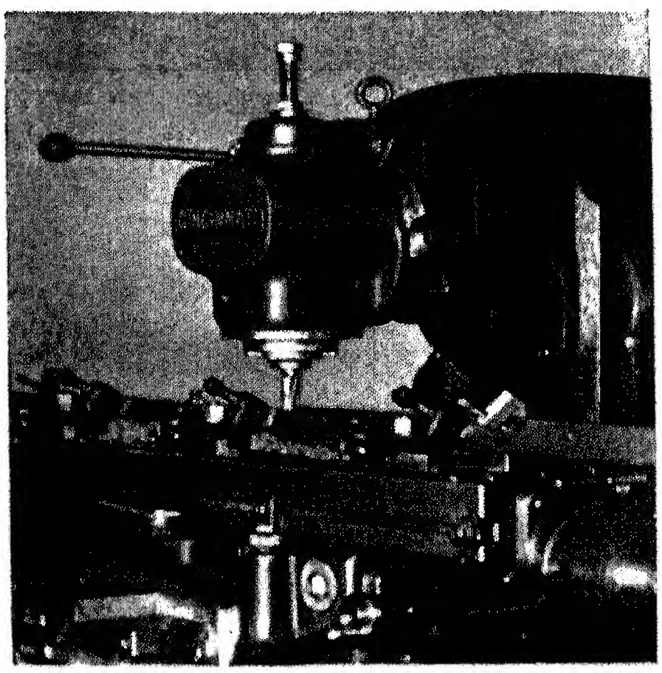


Fig. 13-1. A vertical milling attachment. (Courtesy The Cincinnati Milling Machine Co.)

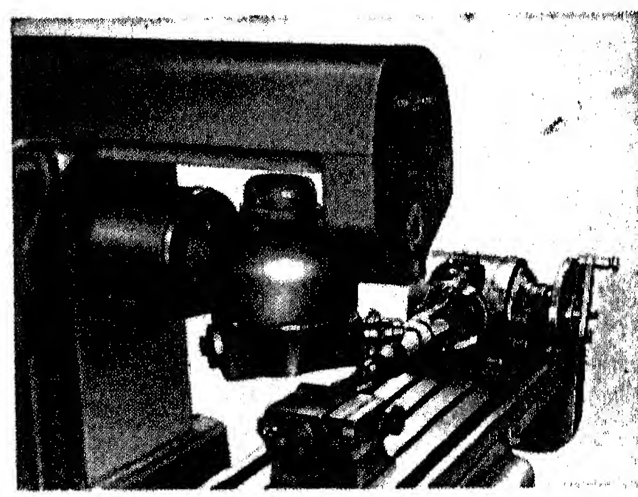


Fig. 13-2. A universal spiral milling attachment set up for milling a large helix angle. (Courtesy The Cincinnati Milling Machine Co.)

mounted at the end of the spindle of either a horizontal or vertical miller and holds the cutter in its normal position on the machine.

The *universal high speed milling attachment* of Fig. 13-3 can be swiveled 360° in a plane parallel to and 45° either way in a plane normal to the machine column face. This attachment is supported by the overarm and is driven by the machine spindle. Some attachments of this type are fully mounted on special overarms and have their own driving motors.

Thread and rack milling attachments. A thread milling attachment is used on universal milling machines for milling threads and similar helices with helix angles larger than 45°. It carries a cutter

on the right-hand end of its spindle but resembles the rack milling attachment of Fig. 13-4 in that both have spindles fixed at right angles to the machine spindle and parallel to the top of the table.

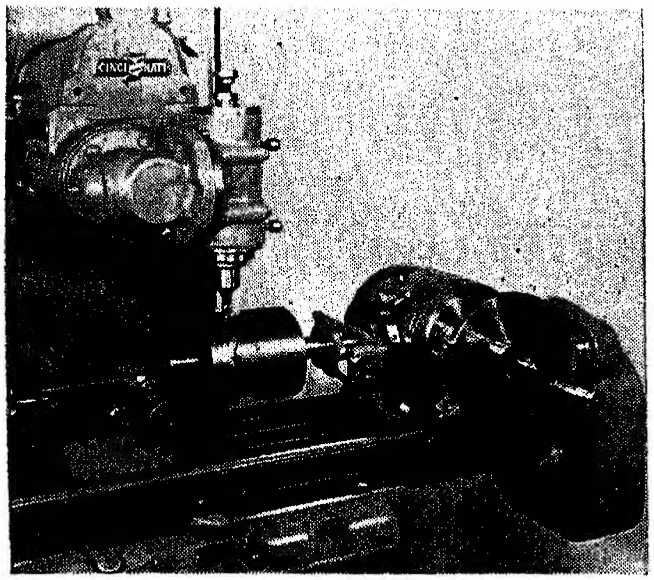


Fig. 13-3. A universal high speed milling attachment and end mill and dividing head with helical milling attachment on a plain knee and column type milling machine for end milling a cam track. (Courtesy The Cincinnati Milling Machine Co.)

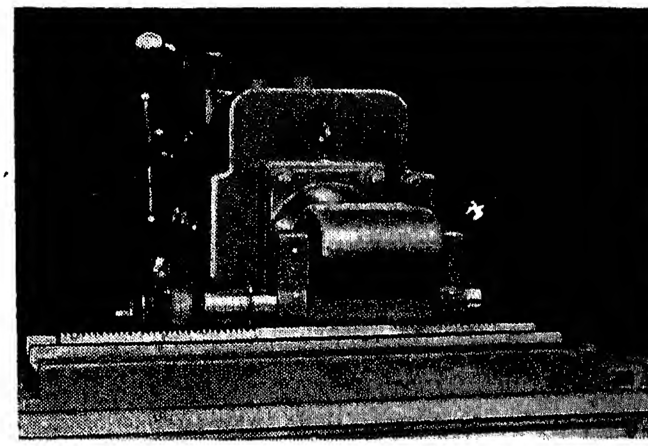


Fig. 13-4. A rack milling attachment. (Courtesy Kearney and Trecker Corp.)

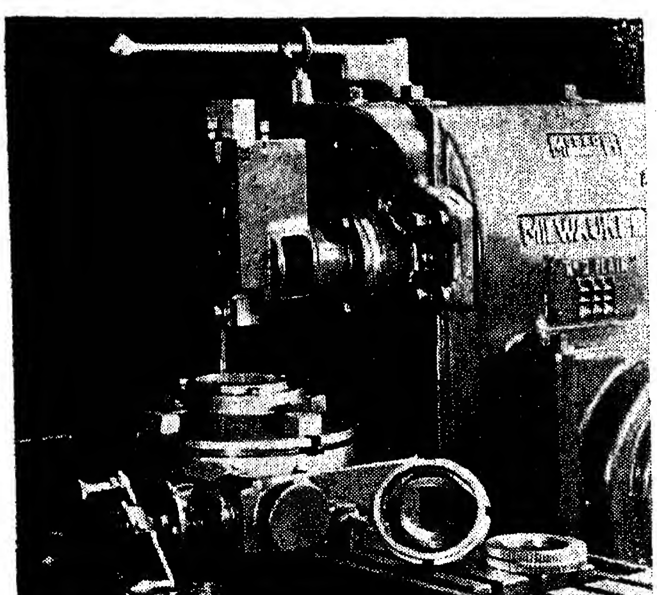


Fig. 13-5. A slotting attachment applied to machining serrations in a workpiece. (Courtesy Kearney and Trecker Corp.)

These attachments may also be used for cross milling and cutting off long pieces.

The machine table may be indexed for milling a rack like the one in Fig. 13-4 by means of the table feed dial or a *rack indexing attachment*. That unit consists of a gear attached to one end of the table leadscrew and in mesh with a train of several other gears. An indexing pin engages the spaces between the teeth of a gear at the end of the train.

Slotting attachment. A milling machine can be adapted to doing the work of a slotter or small shaper for occasionally cutting keyways, internal gears, serrations, etc. by means of the *slotting attachment* shown in

Fig. 13-5. This attachment is driven from the machine spindle at the same number of strokes per minute as the number of spindle revolutions per minute. The slide and ram can be swiveled in a plane parallel to the machine column through 360°, and the length of stroke can be changed from zero to several inches.

Work Holding and Positioning Attachments

Work must be securely held for milling. When one or a few pieces are to be milled, a workpiece may be fastened and located on the table by tee bolts, strap clamps, step blocks, parallels, angle plates, keys, etc., as is done on shapers, planers, and drill presses. Setup in this way is slow and is not even feasible for many kinds of work that can be done on milling machines, such as gear cutting. A number of kinds of devices are widely used to make the mounting of work on milling machines faster and easier to satisfy specific situations. The most common devices will now be described.

Vises, chucks, and fixtures. Vises are the most common holding devices found on milling machines. Four styles are the plain, swivel, toolmakers' universal, and rack vise.

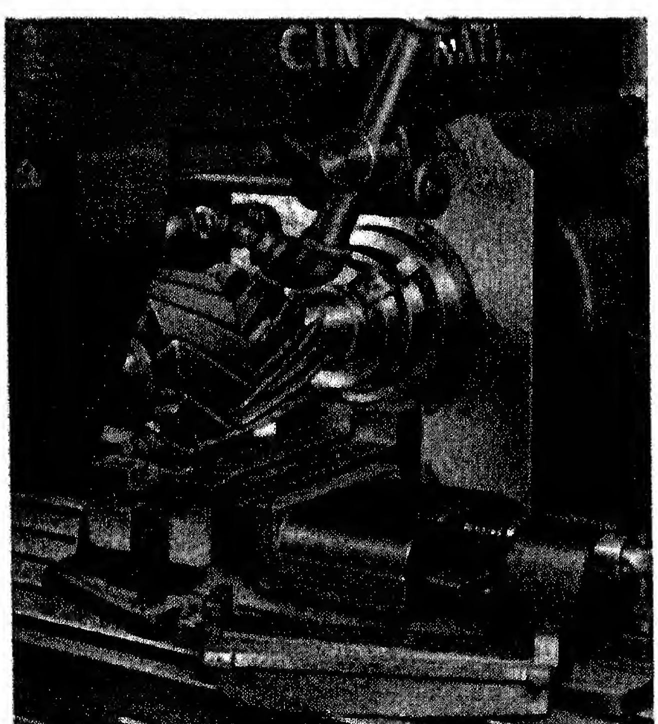


Fig. 13-6. A plain milling vise with special jaws for holding an irregular workpiece. (Courtesy The Cincinnati Milling Machine Co.)

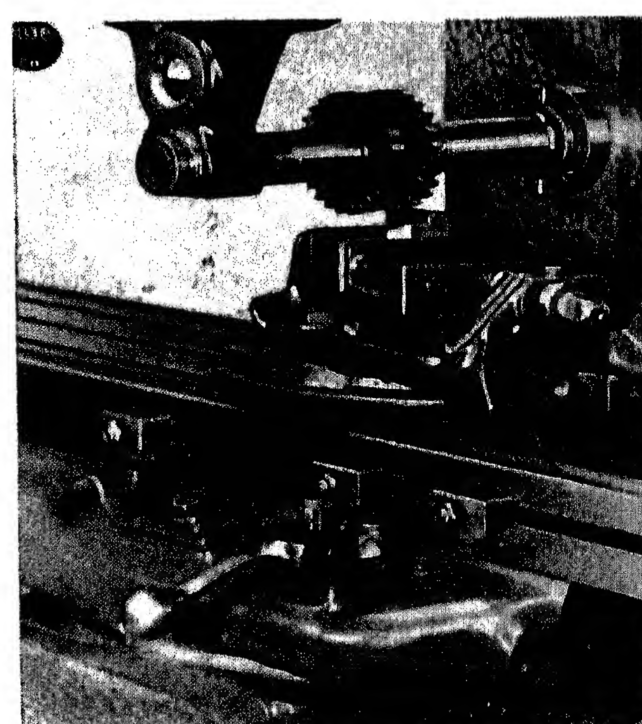


Fig. 13-7. An application of a swivel vise. (Courtesy Kearney and Trecker Corp.)

A *plain vise* is fastened directly to a milling machine table. Two sets of keyways and bolt slots in the base enable the vise to be located from and fastened to the T slots in the table so that its jaws are either parallel or perpendicular to the direction of movement of the table. Plain vises are shown in these two positions in Figs. 12-4 and 13-6. The fixed jaw is part of the body, and the movable jaw slides on and is closely fitted to the body. The most common clamping means is a screw and nut to exert large forces, but faster acting cam clamping arrangements are often used.

The vise jaws that make contact with the work are held on by screws and are removable. Standard jaws for general-purpose work have flat faces. Special jaws of irregular shapes are often made and applied to hold specific workpieces milled in quantities. Such jaws on a standard vise are cheaper than a whole special fixture. An example of such an arrangement is given by Fig. 13-6.

The *swivel vise* has the same body as the plain vise, but it is mounted and can be swiveled on a base as shown in Fig. 13-7. The base is bolted and keyed to the table and has graduations to show how much the vise body is swiveled.

The *toolmakers' universal vise* can be swiveled 360° in a horizontal plane and can also be

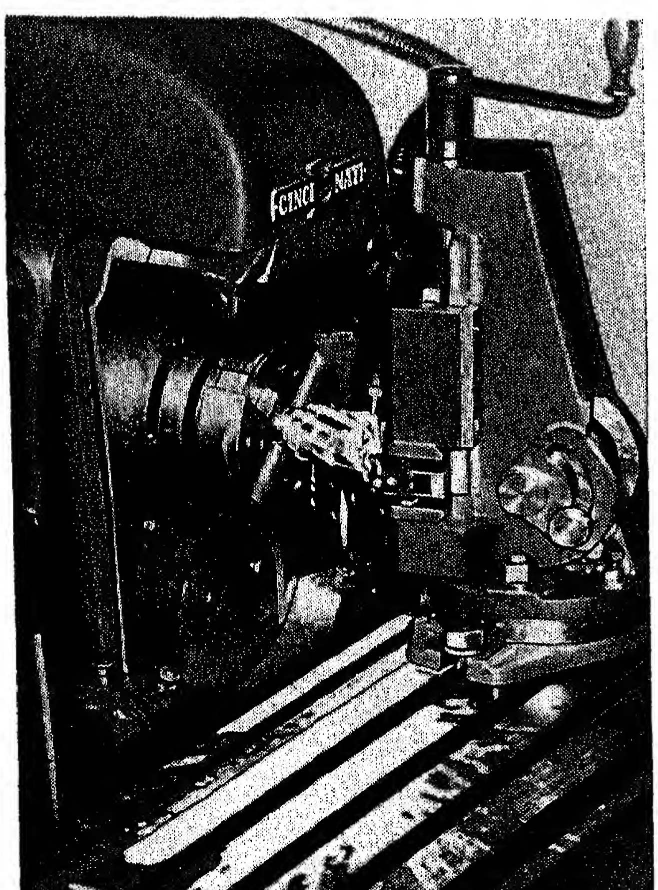


Fig. 13-8. An application of a toolmakers' universal vise. (Courtesy The Cincinnati Milling Machine Co.)

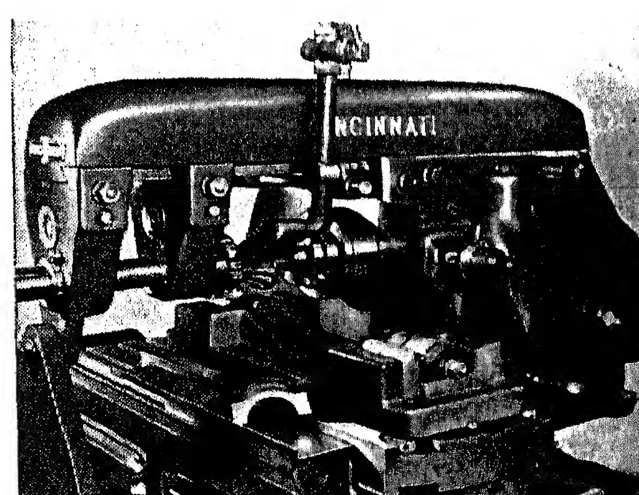


Fig. 13-9. An index base with duplicate fixtures on the table of a plain Hydromatic milling machine. (Courtesy The Cincinnati Milling Machine Co.)

tilted and clamped in any position from 0° to 90° in a vertical plane. Thus compound angles can be milled with it, as exemplified in Fig. 13-8. Because of its necessary flexibility, this vise cannot hold work as rigidly as some others.

A *rack vise* is intended for holding long workpieces lengthwise on a milling machine table. One is shown in Fig. 13-4. A number of clamping screws distributes the clamping force over the length of the jaws, and a series of short pieces can be held at one time.

Universal chucks like those used on lathes are employed to hold work on milling machines. A chuck may be mounted on the end of a machine spindle to rotate a piece that is turned, bored, etc. by tools carried on the table. Chucks are commonly mounted on dividing head spindles to hold work that is to be indexed.

Fixtures are special devices designed to hold work for specific operations more efficiently than standard attachments. A fixture may be made to hold a part in a particular position. A fixture may be built to provide full support to a workpiece to keep it from being distorted when cut or clamped. Fixtures are arranged for unloading and loading work quickly and easily. A fixture usually is applicable to only one operation and is justified only when it can save enough on that operation to pay for its cost. Consequently, fixtures are generally made for parts produced in quantities, although sometimes the saving on each part is large enough for a fixture to be worth while for just a few pieces. Fixtures are shown on the machines of Figs. 12-6, 12-8, 12-9, 12-14, 13-9, and 13-10.

Index base and circular milling attachment. An index base is essentially a swivel table with two stop positions accurately spaced 180° apart. Fixtures are mounted on each end of the attachment as depicted in Fig. 13-9. While pieces at one end are being milled, the operator loads the fixtures at the other end. When the cut is finished, the machine table retracts from the cutters, and the operator swivels the attachment through 180° and clamps it. The operation is then repeated. This is known as *index base milling.*

A circular milling attachment or *rotary table* is bolted to the reciprocating table of a milling machine to add a rotary motion to the three straight line motions. A rotary table may be hand fed only or may be power and hand fed.

A hand-fed rotary table with an index unit for spacing holes, grooves, or slots accurately is mounted on the miller in Fig. 13-5.

Some attachments do not have the index unit, only a crank in the same place for feeding the table. The table has graduations in degrees to show directly the amount it turns.

Power feed for a rotary table is derived from the machine table feed screw drive gear, is transmitted through a long shaft to a bracket on the end of the machine table, and is then carried to the

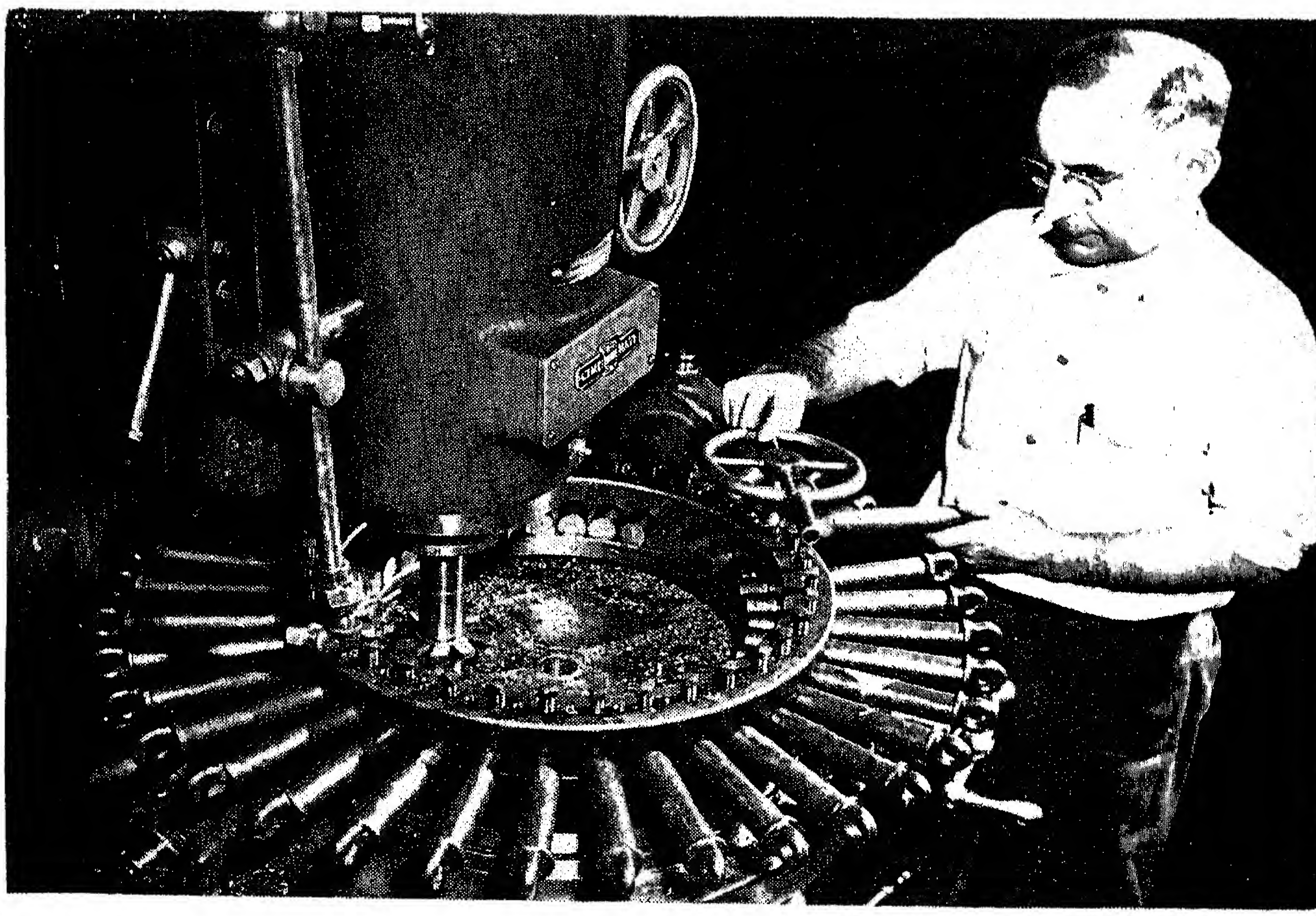


Fig. 13-10. A multiple station fixture mounted on a power driven circular milling attachment for continuous rotary milling. (Courtesy The Cincinnati Milling Machine Co.)

attachment by a shaft along the top of the table. Many curved profiles may be milled by combining the circular motion of a rotary table with the table feed and cross-feed of the machine. If the rotary motion alone is used, fixtures may be put on the table for *rotary* or *continuous milling*, as is done in Fig. 13-10. Workpieces are unloaded and loaded as they pass the operator while others are being cut.

Dividing head and related attachments. A dividing head is a mechanical device for dividing a circle accurately into equal parts. A number of instruments are available for that purpose, but the most

versatile and widely used kind for milling machines is the *universal dividing head* shown in Figs. 12-2, 12-3, 13-2, and 13-3. Its spindle can be tilted at any angle from 5° below the horizontal to 50° past the vertical and clamped in the desired position. A vernier scale measures the angle of inclination. A *plain index head* has a spindle fixed in a horizontal position.

Indexing is the operation of setting off equal divisions on a circle with a dividing head or other suitable device. Common methods of doing this are called direct, plain or simple, compound, and differential indexing.

Direct indexing is done by means of a plate mounted on the spindle of the index or dividing head. The plate has equally spaced notches or holes in one or more circles. The only divisions available are those on the plate. Some indexing devices, such as the plain index head, are capable of doing only direct indexing. They are used where the number and variety of divisions required are small, such as where a few sizes of gears, squares, hexagons, etc. are cut in quantities. Most universal dividing heads have plates for direct indexing but also means for indexing by one or more of the other methods.

Plain indexing is done through a single train of gears. The sections of a universal dividing head of Fig. 13-11 show a worm turned by gears and a crank on the side of the head. The worm is engaged with a worm wheel around the spindle. Forty turns of the crank are needed to turn the work spindle one full revolution. A pin in

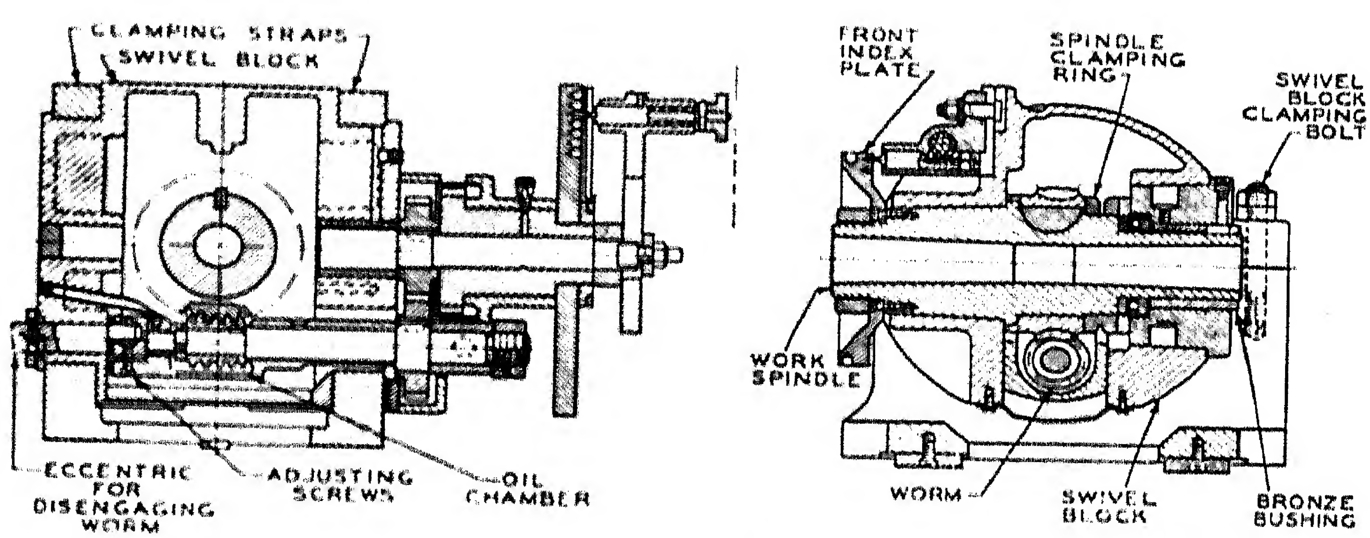


Fig. 13-11. Sections through a universal dividing head to show the indexing mechanism. (Courtesy The Cincinnati Milling Machine. Co.)

the crank handle registers in holes in a plate fixed to the side of the head as also shown in Figs. 12-2, 12-3, 13-2, and 13-3. The holes in the plate are arranged in circles and are equally spaced in each circle. Each successive circle from the center has a larger number of holes. Standard plates on one make of dividing head have holes that make it possible to index all numbers up to and including 60, all even numbers and those divisible by 5 from 60 to 120, and many higher numbers up to 1000. Other plates, which are interchangeable, are available with different numbers of holes.

A popular make of dividing head has a hypoid gear drive between the crank and spindle. Five turns of the crank result in one full turn of the spindle.

On a dividing head with a 40 to 1 ratio, one full turn of the crank is needed to index a 40 tooth gear from one tooth space to the next. One half turn is needed to index an 80 tooth gear from space to space. Also, two turns of the crank are required to index a 20 tooth gear from one space to the next. In general, the number of turns of the crank to index the work one division is equal to the ratio of the dividing head (40 in the example just given) divided by the number of equally spaced divisions required around one full turn of the workpiece.

To index the work a specified amount may require a whole number of turns of the crank, a fraction of a turn like 1/3, 1/7, or 1/15, or a whole number of turns plus a fraction of a turn like 1 1/3, 2 1/7, 3 1/15, etc. A partial turn of the crank is gaged by the holes in a circle on the index plate. The distance between two adjacent holes is called a space. A circle that has 18 holes also has 18 spaces. Two such spaces represent 1/9 of a turn, 6 spaces 1/3 turn, and 9 spaces ½ turn of the crank, and so on. Thus, the number of spaces that represent a specific fraction of a full turn of the crank is equal to that fraction times the number of spaces or holes in the circle.

As an example, a dividing head with a 40 to 1 ratio is to be used to cut a gear having 36 teeth. The crank must be turned $40 \div 36 = 1\frac{4}{36} = 1\frac{1}{9}$ turns to index from one space to another on the gear. If the index plate has a circle with 18 holes, the crank must be advanced $18 \times 1/9 = 2$ spaces after each full turn. Thus, to index from one tooth space to another, the crank is turned one full turn and two spaces.

Actually the spaces do not have to be counted each time the crank

is turned on a dividing head. Two *sector arms* or fingers are mounted on the face of the side index plate. The sector arms can be adjusted to set off any desired number of spaces and serve as a guide for turning the crank.

The dividing head has been called a jewel among machine tools because of its precision. Plain indexing is commonly done on standard heads with an error from hole to hole on the work of less than one minute of arc. That is equivalent to 0.0015 in. on the circumference of a 12 in. diameter circle or one part in 25,410.

Compound indexing is done by engaging a fixed pin in one circle and the crankpin in another circle of holes in an index plate. The plate is turned forward or backward a definite number of spaces each time the crank is turned. This method has been used for numbers not obtainable by plain indexing but has largely been displaced by other forms of indexing.

Differential indexing is used for numbers not readily available from plain indexing, especially large numbers. A typical arrangement is shown in Fig. 13-12. The spindle of the dividing head is geared to a jack shaft that turns the side index plate. Thus when the crank is turned and causes the spindle to turn, the plate is moved forward or backward at the same time. If the crankpin is moved from one hole to another on the plate, the actual distance

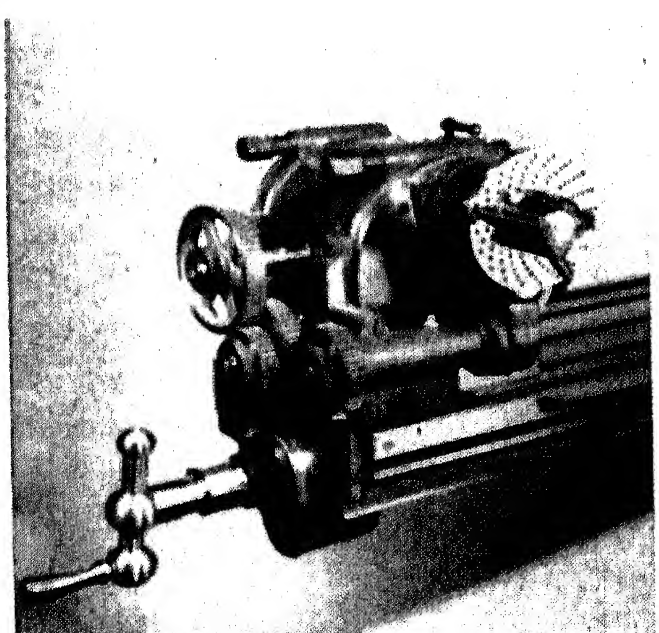


Fig. 13-12. A dividing head set up for differential indexing. (Courtesy Brown and Sharpe Mfg. Co.)

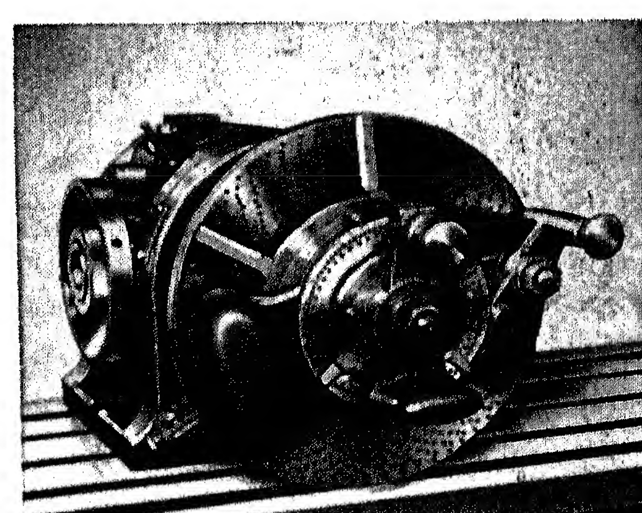


Fig. 13-13. A dividing head with a wide range divider. (Courtesy The Cincinnati Milling Machine Co.)

the crank moves depends upon the position reached by the second hole as the plate moves. The movement of the plate is determined by the gear ratio between the spindle and jack shaft.

The *wide range divider* on the dividing head in Fig. 13-13 has two plates. The long arm engaged with the large plate turns the spindle at the usual ratio of 40 to 1. The small crank furnishes an additional ratio of 100 to 1 through a differential mechanism and is engaged with a plate having a 100 hole circle and a 54 hole circle. The combination of these two ratios provides indexing for 2 to 400,000 divisions, at angular intervals as small as 6 seconds. Another form, the *astronomical dividing head attachment,* has three plates and cranks arranged to divide a circle into degrees, minutes, and seconds.

A *helical milling* or *lead attachment* is mounted on the end of a milling machine table, as in Figs. 12-2 and 13-3, to provide a drive from the table leadscrew to the dividing head to cut helical gears, worms, threads, twist drills, etc. The attachment drives the jack shaft that is geared to the index plate of the dividing head. The crank is engaged with and turns with the plate, causing the spindle to revolve. The lead of a helix cut on a milling machine is equivalent to the distance the table advances while the dividing head spindle makes one full revolution. The lead is varied by means of change gears in the driving attachment. One conventional attachment provides leads from 0.670 to 149 in. Others are available for leads as low as 0.010 in. and almost as high as 3000 in.

Precision measuring attachment. Holes and other surfaces can ordinarily be spaced on a milling machine by use of the graduated dials on the leadscrew and cross screw. The dial graduations indicate 0.001 in. increments of movement and are correspondingly reliable on new machines of good quality. That accuracy is impaired by wear and usage. For more accurate results, a *precision measuring attachment* is applied and makes use of micrometer rods placed between a table or saddle stop and indicator in the same way as described for the jig boring machine in Chapter 14.

Milling Machine Operations

Setup and operation. A rigid and secure setup is necessary for a successful milling operation because the forces that act are rela-

tively large. Adapters, arbors, and cutters must be securely fastened to the spindle. Those that fit into the tapered spindle hole are usually drawn in tightly with a draw bar. Large face mills are bolted to the spindle nose. Small cutters with shanks having sticking tapers may just be inserted in adapters, and those with straight shanks may be gripped in chucks, but care must be taken that they are seated fully. A small cutter may be tapped with a soft hammer to make it secure in its adapter. Cutters, except for frail saws and form cutters, are keyed to arbors on which they are mounted. The arbor nut should be drawn up tightly, but should never be tightened or loosened with a wrench unless the arbor is fully supported, otherwise the arbor may be bent. The overarm and arbor supports must also be clamped tightly.

Cutters of all kinds should be mounted with as little overhang as possible to promote rigidity. A cutter on an arbor is preferably placed near the spindle nose. If the cutter must be placed farther out on the arbor, it should be near the arbor support. Where more than one cutter is carried on an arbor at intervals, two or more arbor supports may be used, as in Fig. 12-8.

Another important consideration in a milling operation is to avoid runout of the cutter. All mating surfaces such as the spindle hole and nose and tapered shanks should be cleaned carefully and have nicks and burrs stoned off before being assembled. A chip or piece of dirt between two surfaces can very easily cause a cutter to run out appreciably. The end faces of arbor collars, especially, should be cleaned carefully. An arbor can readily be bent out of line if dirty collars are drawn together on it. Care must be taken that cutters are sharpened uniformly all the way round so that they run as true as possible.

The work must be held securely on a milling machine, as well as the cutters, to resist deflection and chatter. A workpiece must be well clamped and adequately supported. The workpiece must be kept as close to the table as possible, and cutting forces must be directed against the fixed portions of the work-holding device and the machine. With a vise, the milling thrust should be directed against the solid jaw rather than the movable one. A piece must be clamped down all around in a fixture, so that no part of it is free to spring. Weak sections are reinforced with jack supports.

For quite accurate work, a line or surface on a part, a fixture, or

a vise may have to be aligned with the movements of a milling machine. A common procedure for doing this can be exemplified by a workpiece that has a face that must be aligned with the longitudinal and vertical movements of the table. An indicator is attached on an arm to a convenient stationary rest such as an arbor support, and its spindle is brought in contact with the work surface. The table and knee are then moved, back and forth and up and down, and the work is shifted until the indicator shows a steady reading as it traverses the surfaces. Shims may be placed under the work to align the face vertically.

After the work and cutter have been mounted on a milling machine, they must be positioned with respect to each other to locate the cut. If a surface is to be cut in relation to another on a part, a piece of tissue paper is held on the reference surface, and the cutter and work are brought together until the paper is just cut. The work and cutter are then moved apart in a direction that does not change the setting that has just been established. After they are clear, the depth or position of cut can be reached by means of the graduated dials on the machine. A dial must be turned consistently in the same direction to avoid the effects of backlash. Care must be taken that a milling cutter is not placed in a position to be fed into a vise, fixture, or the machine table to cause damage. Fixtures commonly have attached set blocks to which cutters are set by means of feeler gages.

A cutter should be centered with the workpiece to cut gear teeth

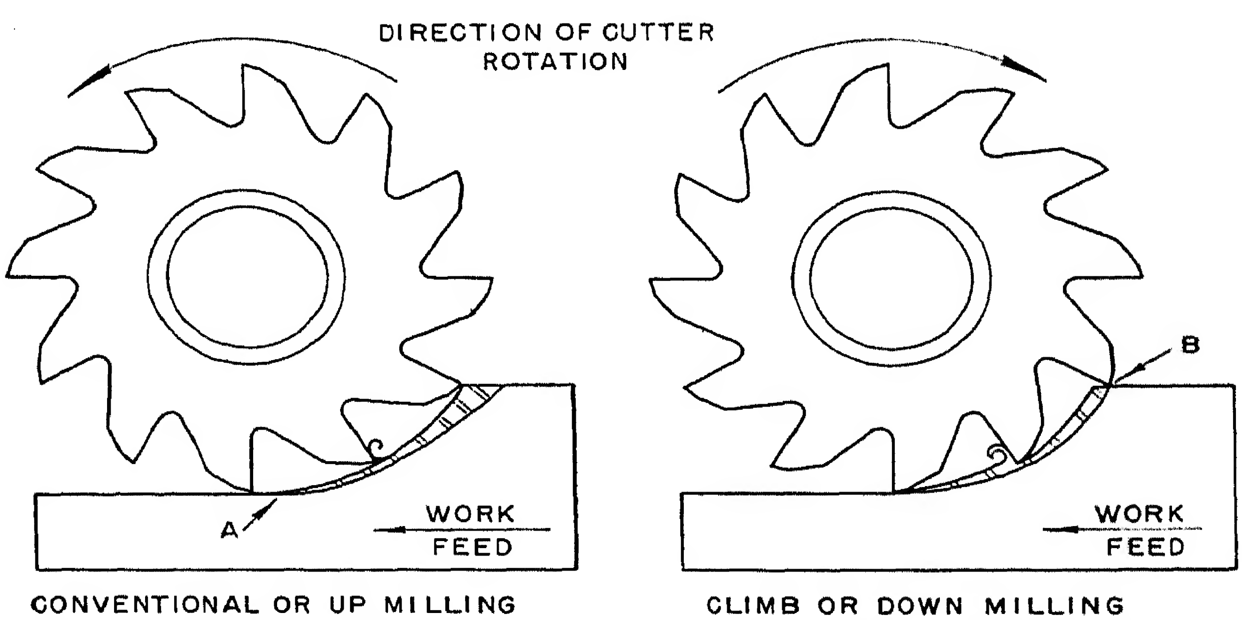


Fig. 13-14. The difference between conventional and climb milling.

or a keyway. If the cutter is made to touch the very top of a round piece, it will clean up a small spot that marks the center of the piece. That mark is the reference for setting the cutter. If the piece is held between centers, the cutter may be positioned with reference to the point of one of the centers.

When a number of pieces is to be milled and provision is made to locate each in the same place, dogs are set along the machine table to trip the feed at the beginning and end of each cut.

Tolerances of less than 0.001 in. can be held when milling a few pieces at a time with sufficient care and skill. However, tolerances of 0.002 to 0.005 in. are more practicable and economical for production milling. Cemented carbide milling cutters run at high speeds are preferred for milling to small tolerances because they produce relatively little heat in the workpiece, but cause most of the heat to be carried off with the chips.

Conventional or up milling as depicted on the left of Fig. 13-14 is common practice for slab or groove milling. The action of the cutter is opposed by the feed of the work. A tooth starting to cut has little material to remove. It tends to slide over the work surface at first until pressure builds up enough to force the edge into the material at a place such as A in Fig. 13-14. That is hard on the cutting edge and leaves pronounced feed marks on the work surface.

In *climb* or *down milling* a tooth enters the work with a substantial bite as shown at B on the right of Fig. 13-14. The cut is cooler, cutters last longer, feed marks are reduced, and the work is held down. However, the cutter tends to pull the workpiece along. The average milling machine has too much backlash in the feed screw and nut and allows the cutter to draw the workpiece ahead and take bites that are too large. Damage is likely to result. Thus, climb milling must never be attempted unless the machine, workpiece, and holding device are suited for it. Backlash is eliminated on hydraulic and some mechanical milling machines built especially for climb milling. The workpiece and holding device must be strong enough to resist the forces present in climb milling.

Speed, feed, and depth of cut in milling. Substantially the same factors determine economical speeds for milling cutters as for single point tools. No rules can be given that apply to all cases. The average surface speeds given in Table II of Chapter 4 may be used for milling cutters in the machine tool laboratory. Recommenda-

tions for cutting speed for milling cutters under many different conditions are given in handbooks.

A spindle speed in rpm on a milling machine is selected to give the desired surface speed on the periphery of the cutter in sfpm. The spindle speed in rpm is approximately equal to 4 times the cut speed in sfpm divided by the diameter of the cutter in inches. The relationship is the same one that determines the turning speed of a workpiece on a lathe.

The logical basis for feed in milling is the distance the work advances in the time between engagements by two successive teeth. This is called feed per tooth in units of inches per tooth or ipt. However, the machine feed rate is given in inches per minute and is equal to the feed in ipt times the number of teeth in the cutter times the number of rpm of the cutter.

Feed per tooth should be as high as possible for economical milling. The higher the feed per tooth, the fewer times the teeth must contact the work to remove a given amount of stock. On the other hand, the heavier the feed per tooth, the greater the load on the cutter teeth, workpiece, holding device, and machine. A large face mill will withstand a greater feed per tooth than a small end mill. A light feed may have to be chosen for a fragile workpiece. The rigidity and power of a milling machine may limit the rate at which stock can be removed. A heavier feed is possible in soft materials than in hard or tough metals. Practical feed rates in ipt are given in handbooks for many situations. The following average feeds for HSS cutters may be used as a general guide.

Table VI

Feeds in ipt for average conditions

Material	*Face mill*	*Helical mill*	*Slotting and side mills*	*End mill*	*Form relieved cutter*	*Slitting saw*
Cast iron (medium)	0.013	0.010	0.007	0.007	0.004	0.003
Steel (low carbon)	0.010	0.008	0.006	0.005	0.003	0.002

Feeds are generally at least 20 per cent higher for carbide cutters. On light finishing cuts, the feed may be reduced to improve the surface finish.

If the amount of stock varies considerably on a rough workpiece, the forces and cutting action can be expected to vary, and the surface milled may not be true. One cut is enough for most jobs, but

rough and finish cuts are required to produce the best surface finishes and exceptionally accurate dimensions. The depth of a roughing cut should be as much as the cutter, machine, and work will stand. Less than 1/16 in. is left for finishing.

Not infrequently a milling cutter sets up vibrations in the work and machine because of the periodic action of its teeth. This tendency can be corrected by using a cutter with a different number of teeth or by changing the cutter speed or feed to change the frequency induced by the cutter action.

Planning for economical milling. Setup often accounts for most of the time to mill one or a few pieces of a kind. Much time can be saved where a variety of work is milled by planning and scheduling the jobs so that similar parts are milled in succession. When all or some of a setup for one part can be used for another, tear-down and setup time is saved.

Simple milling involves the loading and milling of one piece at a time. When a quantity of pieces is to be milled, other arrangements are able to save time. For *string or line milling*, two or more pieces are held in a row. Cutting time is saved because the cutter can be entering one piece while it is leaving another. Savings can be realized by arrangements to allow the operator to load at one station while the machine is cutting at another. One such arrangement is called *reciprocating milling* and employs fixtures at both ends of the table, as in Fig. 12-4. The operator loads one station while work in the other is being milled. Other arrangements for the same purpose are *index base* and *rotary milling*, already described in connection with the attachments used.

Milling compared with other operations. Milling machines are capable of doing a large variety of work. Single plane surfaces are machined by plain, slab, or face milling, and multiple planes by straddle and gang milling, slotting, etc. Irregular and curved surfaces are machined by profiling, duplicating, and form, angular, cam, and template milling. External round surfaces may be turned and holes may be drilled, bored, and reamed on milling machines.

Work is revolved only occasionally on a milling machine because lathes are more efficient for such operations. A drill press is easier to set up and operate than a milling machine for holes that do not need to be located accurately. When holes are to be located with a fair degree of accuracy, the milling machine is often the best

choice for small quantities. More accurate location can be obtained on jig boring machines. For large quantities, the milling machine is usually slower and not able to compete with the use of jigs on drilling machines or with boring machines. Really large pieces require the capacity and range of horizontal boring machines, beyond that of most milling machines.

Flat, straight, and many curved and irregular surfaces can be machined on shapers and planers as well as on milling machines. The advantages of shapers and planers, particularly for one or a few pieces, were pointed out in Chapters 9 and 10. In general, setup and cutter sharpening time and the cost of the equipment are larger for a milling machine than for a shaper or planer. In spite of this, the milling machine is usually more economical for moderate and large quantities of parts because it can remove metal faster.

Broaching is more economical than milling in many cases where large quantities of parts are produced. However, many parts that can be milled cannot be broached. For instance, a pocket in a piece may be accessible for milling but may not be readily broached. Many parts that are strong enough to withstand the cutting forces of milling are too weak for the larger forces of broaching. Where broaching is feasible, it usually is faster than milling. Although a broaching machine is simple and often less expensive than a comparable milling machine, the tooling required for each job on a broaching machine is much more costly. However, broaching tools have long lives, and if production requirements are high and continuous, the unit cost is lower for broaching than for milling.

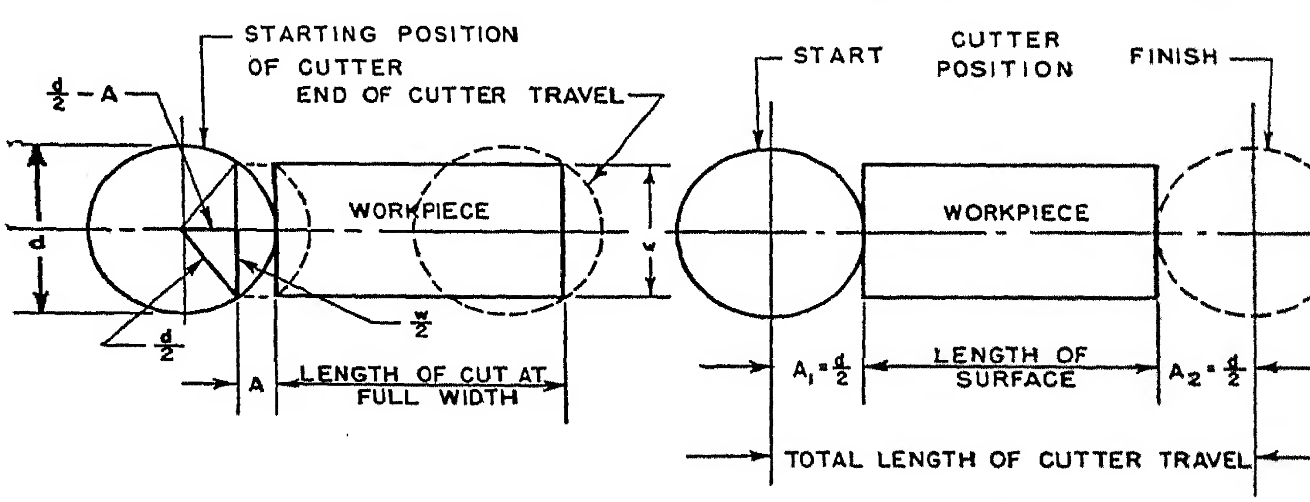


Fig. 13-15. A. Approach of face mill for roughing cut. B. Approach of face mill for finishing cut.

Grinding is capable of producing closer tolerances and finer finishes than milling. However, grinding is not generally an economical means for removing much stock, and surfaces are commonly milled before being ground.

Estimating milling time and power. The approach of a milling cutter may be an appreciable part of the length of cut. A face mill taking a roughing cut is normally stopped when it has just cleaned the surface as indicated by Fig. 13-15 A. The approach shown at the beginning of the cut can be calculated from the right triangle having sides $d/2 - A$, $w/2$, and $d/2$. It is $A = d/2 - \sqrt{(d^2 - w^2)/4}$. However, if the diameter of the cutter is only a little larger than the width of the surface, the approach is $d/2$ for practical purposes. For a finishing cut, a face mill is passed entirely over a surface so that its trailing edge can get in a full wiping action. Thus, for finishing the approach is equal to d.

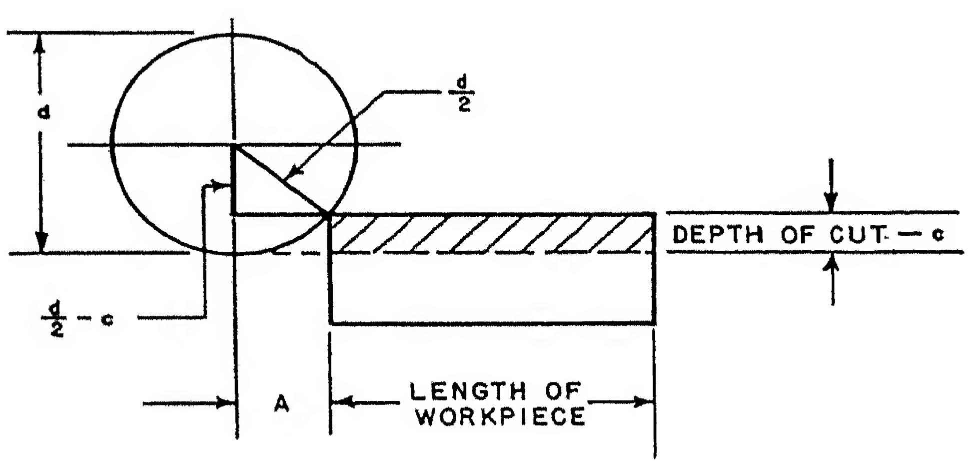


Fig. 13-16. Approach of plain or slot milling cutter.

The approach of a cutter milling a flat surface or a slot occurs from the first contact until full depth of cut is reached, as depicted in Fig. 13-16. It can be calculated from the right triangle as shown and is $A = \sqrt{cd - c^2}$. Tables of milling approach are found in handbooks.

The total length of cutter travel is the sum of the overtravel, the approach, and the length of the workpiece surface.

The rate of metal removal in cubic inches per minute for milling is the product of the width of cut in inches times the depth of cut in inches times the feed in inches per minute. The horsepower required at the cutter is equal to the product of the unit power con-

sumption in horsepower per cubic inches per minute times the rate of metal removal in cubic inches per minute. Some average values of unit power consumption for milling are given in Table IV on page 185.

Questions

1. What benefits do milling machine attachments give?
2. What does a vertical milling attachment do for a milling machine?
3. What are the uses of a universal milling attachment, a high speed milling attachment, and a slotting attachment?
4. Describe the common ways of holding work on a milling machine.
5. What advantages may be derived from fixtures?
6. Describe index base milling and circular base milling. For what purposes can each be used?
7. Name and describe the four common methods of indexing.
8. Describe a helical milling attachment. For what is it used?
9. What is the difference between conventional or up milling and climb or down milling? What are the advantages and disadvantages of each?
10. How should jobs be scheduled when a variety of milling is done?
11. How may cutting time and loading time be saved in milling operations?
12. When is milling preferred over shaping or planing?
13. Compare milling and broaching.

Problems

1. Specify the number of turns, the number of holes in the circle, and the number of spaces to index the following divisions on a universal dividing head with a 40 to 1 ratio.
   (a) 120 (b) 100 (c) 96 (d) 75 (e) 48 (f) 34 (g) 30 (h) 26 (j) 24 (k) 18 (l) 15 (m) 13 (n) 9
2. Write an operation instruction sheet for machining one piece shown in Fig. 9–16 on a milling machine. The rough casting has ⅛ in. stock on all surfaces. Normal operation tolerances are satisfactory.
3. A lot of 40 pieces shown in Fig. 9–16 is to be milled. Write a route sheet describing the operations required to finish each part. Write an instruction sheet for each operation.
4. An operation is to be performed to machine the sides and bottom of the dovetail grooves on a number of parts of the kind shown in Fig. 9–16. Setup requires 20 minutes on a shaper and 30 minutes on a

milling machine. The direct time to machine each piece on the shaper is 14 minutes, and on the miller it is 11 minutes. Labor costs \$1.50 per hour. The charge for the use of the shaper is \$3.00 per hour, and for the milling machine \$3.50 per hour.

(a) What is the quantity below which the shaper is more economical and above which the miller is more economical?

(b) Which machine would you choose for 5 pieces? For 25 pieces?

Chapter 14

BORING MACHINES AND OPERATIONS

SEVERAL KINDS OF MACHINE TOOLS are called boring machines because they are used largely for locating and finishing holes. Most of them also do allied operations like drilling, reaming, threading, and facing, and some do turning and milling. Outside of boring, many machines in this category have little in common with each other. Some are horizontal, others vertical; some large, others small; some are intended for general-purpose work, others for special jobs in large quantities; one may have a single spindle, another many spindles; etc. Four commonly recognized classes are horizontal boring mills, vertical boring and turning machines, precision boring machines, and jig boring machines. The machines in these groups are the ones to be presented.

Horizontal Boring Mills

Horizontal boring mill is the common name for the *horizontal boring, drilling and milling machine.* Machines of this class do drilling, boring, reaming, turning, threading, facing, milling, and other operations on workpieces that are too bulky, irregular, unsymmetrical, or heavy to be rotated conveniently or even at all. The standard forms of this machine are the table, floor, portable, planer, and multiple head types.

Table-type horizontal boring mill. The table type is the most versatile and common horizontal boring mill, so named because work is carried and moved in two coordinate directions on a horizontal table. The heavy cylinder block on the large horizontal boring mill of Fig. 14-1 is carried on a small auxiliary table in ad-

dition to the main table. Most machines have only one table that moves crosswise on a saddle. The table has T slots for clamping bolts. The saddle slides on longitudinal ways on a massive bed. Two bed ways are common, but additional outer ways are provided on machines with extra table cross travel.

The headstock or spindle head is carried on ways on a vertical

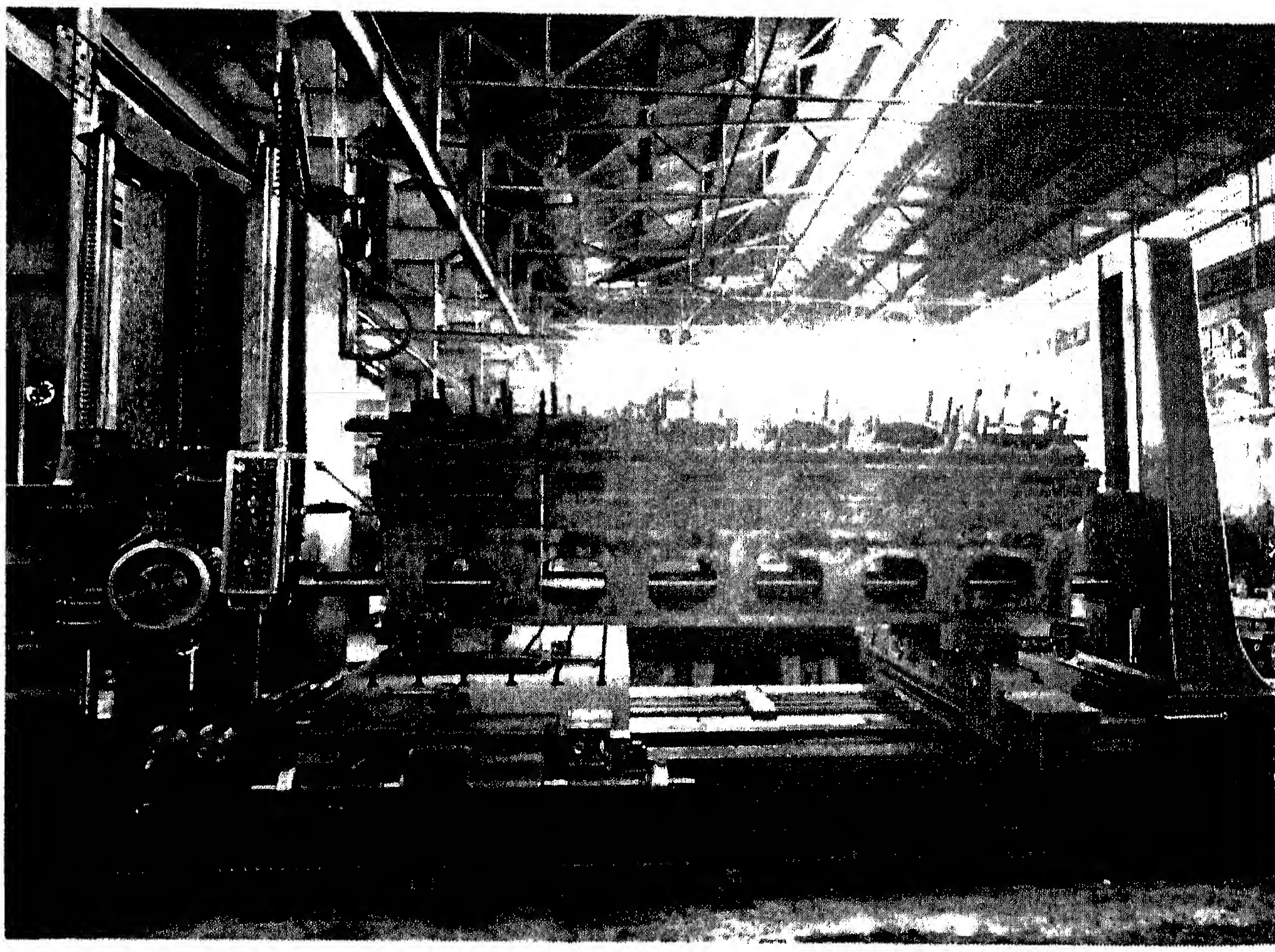


Fig. 14-1. A 6 in. table-type horizontal boring, drilling, and milling machine with an auxiliary saddle and table. A boring bar is arranged between the spindle and backrest block for boring the bearings in a large cylinder block. (Courtesy Lucas Machine Div., The New Britain Machine Co.)

column bolted on one end of the bed of a horizontal boring mill, is counterweighted, and can be moved up or down. The cutter spindle revolves in a quill or sleeve and can be fed in or out of the headstock. Most machines have one spindle, but some large ones have two. One is a light high speed spindle, the other a slow spindle for heavy work.

Some large machines have a feed drive motor in the bed and a

separate spindle drive motor on the headstock. Most table-type horizontal boring mills have a single drive motor and speed and feed change gear boxes in the bed below the column. A typical medium-size horizontal boring mill has 27 spindle speeds with a range of 12 to 1200 rpm and a 15 hp motor.

The spindle can be moved in or out of the headstock, the headstock raised or lowered, the table moved crosswise on the saddle, and the saddle moved lengthwise on the bed of a table-type horizontal boring mill. The table, saddle, and headstock may be positioned by accurate leadscrews and micrometer dials. Sometimes vernier scales along the ways are provided for setting the machine units. Some machines are equipped with end measuring rods and dial indicators like jig borers for precise settings. These may be combined with electrical switches to stop the feeds.

A typical medium-size horizontal boring mill has power feed rates for the head, table, and saddle from 0.045 to 50 in. per minute and a rapid traverse rate of 125 in. per minute. The spindle can be fed at from 0.004 to 0.125 in. per revolution. The various units may be fed independently or at the same time. Power feeds and rapid traverse are engaged by levers on many machines. Typical control levers are shown in Fig. 14-2. Some modern machines have pendant controls like the one in Fig. 14-1 that can be positioned conveniently for the operator. Buttons on the pendant make it possible to jog, start, quickly stop, or reverse the spindle, and to inch, continuously feed, or rapidly traverse the spindle, table, saddle, or head in forward or reverse directions.

A horizontal boring mill spindle has a tapered hole in its end to take the shanks of cutters and boring bars. Morse tapers are most common. A cross slot at the small end of the tapered hole takes the tang on a cutter or boring bar shank and gives access to a drift to drive out the tool shank. Some spindles have keyways on their ends. A large cutter, like a face mill, usually has a central hole or counterbore that fits the periphery of the spindle snugly.

Long and heavy boring bars cannot be supported by the headstock spindle alone. An outboard support is provided by a backrest or end support shown in Fig. 14-1. This is a column clamped to ways and having a block that is raised or lowered by a screw in unison with the headstock. A bushed hole in the block in line with the machine spindle takes the outer end of the boring bar.

Floor-type and portable horizontal boring mills. Floor-type and portable boring, drilling, and milling machines are designed for workpieces that cannot be accommodated readily on table-type machines because of their weight, size, or shape. The column that carries the headstock of a floor-type machine is mounted on a base that slides on runways. A workpiece is placed on floor plates alongside the runways. Whereas the work is moved past the spindle of a table-type machine, the spindle is traversed on the runways past the workpiece by a floor-type machine. The runways may be made long enough to take in as large an area as desired, and sometimes more than one column and head are carried on a single set of runways.

A portable horizontal boring mill consists of a column and headstock on a base. The entire machine is picked up by a crane, carried to the work, and placed in a convenient position for the job to be done. Portable boring mills are essential for large assemblies like huge turbo-generators and vessels.

Planer-type and multiple head horizontal boring mills. The planer-type horizontal boring mill resembles the table-type, but its table rides directly on the bed instead of on a saddle and reciprocates at right angles to the spindle. The headstock and column and the end support are movable on runways toward and away from the table.

The planer-type machine has a long table fully supported by the bed in all positions. It is capable of giving full and rigid support to long and heavy workpieces.

The multiple head horizontal boring, drilling, and milling machine resembles a double housing planer or a planer-type milling machine. A long table reciprocates and is fully supported on a bed. A column is attached at about the middle of each side of the bed. An adjustable height cross rail bridges the columns. The machine may have 2, 3, or 4 headstocks. One or two are mounted and adjusted along the cross rail. The others are carried on the columns and can be moved up or down.

The multiple head boring machine can do work on two or three surfaces at one time.

Sizes of horizontal boring machines. The size of a horizontal boring mill is designated by the diameter of its spindle in inches.

The proportions of the other units of the machine are related to the size of the spindle. Sizes commonly range from 3 to 7 inches.

A typical 4 in. table-type horizontal boring mill has a 4 in. diameter spindle with a No. 6 Morse taper socket and a 40 by 72 in. table. The longitudinal movement of the spindle is 36 in., the table moves 60 in. at right angles to the spindle, and the spindle may be raised to 48 in. above the table. The machine weighs 3600 lbs.

Tools and attachments. Drills, reamers, taps, and milling cutters of all kinds and sizes are commonly used on horizontal boring mills. Boring bars range from small stub bars to those equal in diameter to the spindle of the machine and several feet in length, to reach from the headstock to the end support at the end of the bed. Boring bars may be piloted in work-holding jigs or in supports fastened to the table instead of in the end support bearing.

A boring bar must be smaller in diameter than the hole through which it must go, but still large enough so that the cutter bit does not have to extend excessively and lack adequate support. A heavy boring bar is shown in operation in Fig. 14-1. For a bore diameter much larger than a feasible size of bar, a boring head may be mounted on the bar. This is a block that carries one or more bits. A boring head may have its own shank and be mounted directly on the machine spindle instead of on a boring bar.

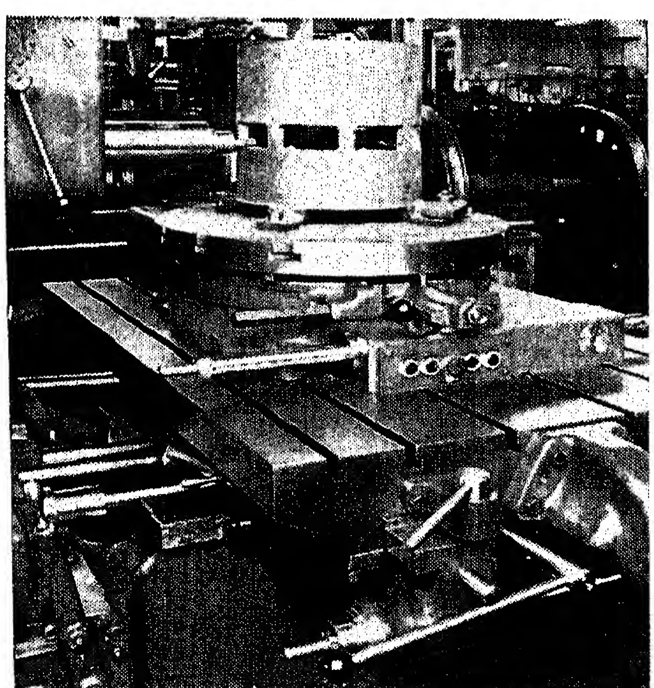


Fig. 14-2. Windows being end milled in a casting mounted on a power operated circular rotary table on a 3 in. table-type horizontal boring, drilling, and milling machine. (Courtesy Lucas Machine Div., The New Britain Machine Co.)

Single point tools, called fly cutters, are most commonly used on boring bars for general-purpose work. A bar may have one or more cutters. Several cutters may take successive rough and finish cuts in one bore or may be arranged to machine a number of diameters and faces in line in one setup. Single point tools are relatively easy to sharpen and adjust accurately. Double cutters extend in line from opposite

sides of a bar. They cut faster than single tools but are less flexible.

The *star feed facing head* and *the continuous feed facing and boring head* are spindle attachments that revolve on the machine spindle and have a single point cutter that is fed outward to make facing or recessing cuts. The tool is fed intermittently by the star feed facing head and continuously by the continuous feed head.

Horizontal, vertical, and tilting rotary tables are used for many operations on horizontal boring machines. A workpiece is mounted on a power-operated circular rotary table in Fig. 14-2. Although the platen shown is round, some are square or rectangular. A power-operated auxiliary table may be rotated continuously at feed or rapid traverse rates or indexed to desired positions. Many tables are manually operated only. Rotary tables range in size from 24 in. diameter hand-fed units to 10 ft round or square power-driven units. A rotary table facilitates machining a number of surfaces and holes at various angular positions on a workpiece without unclamping and resetting the piece for each surface or hole.

Vertical Boring and Turning Machines

A vertical boring and turning machine is like a lathe set on the end of its headstock so that the face plate or chuck is in a horizontal position. The face plate then becomes a table. These machines do essentially the same work and use similar or the same tools as lathes but offer distinct advantages for some kinds of work. Large and heavy workpieces can be laid on the table of a vertical boring machine and positioned with relative ease. A workpiece does not overhang the end of the spindle as it does on a horizontal lathe, but its weight is absorbed directly into the base of the machine. The operator can stand close to the tools and observe their actions while manipulating the controls.

The size of a vertical boring machine is designated by the diameter of its table, and that is nominally the largest diameter that can be swung on the machine. The smaller machines usually have a turret above the table to hold a number of tools for repetitive work and are called *vertical turret lathes.* Their common sizes are 30, 36, and 42 in., but some are as large as 100 in. Without turrets, the machines are called *vertical boring mills.* Their sizes overlap those

of the large vertical turret lathes and range up to above 20 ft in diameter. In addition to the basic single work spindle machines, multiple spindle machines have been developed on the same principles for large quantity production.

Vertical turret lathe. A 36 in. vertical turret lathe is shown in Fig. 14-3 with a piece chucked on the table and tools set up on the

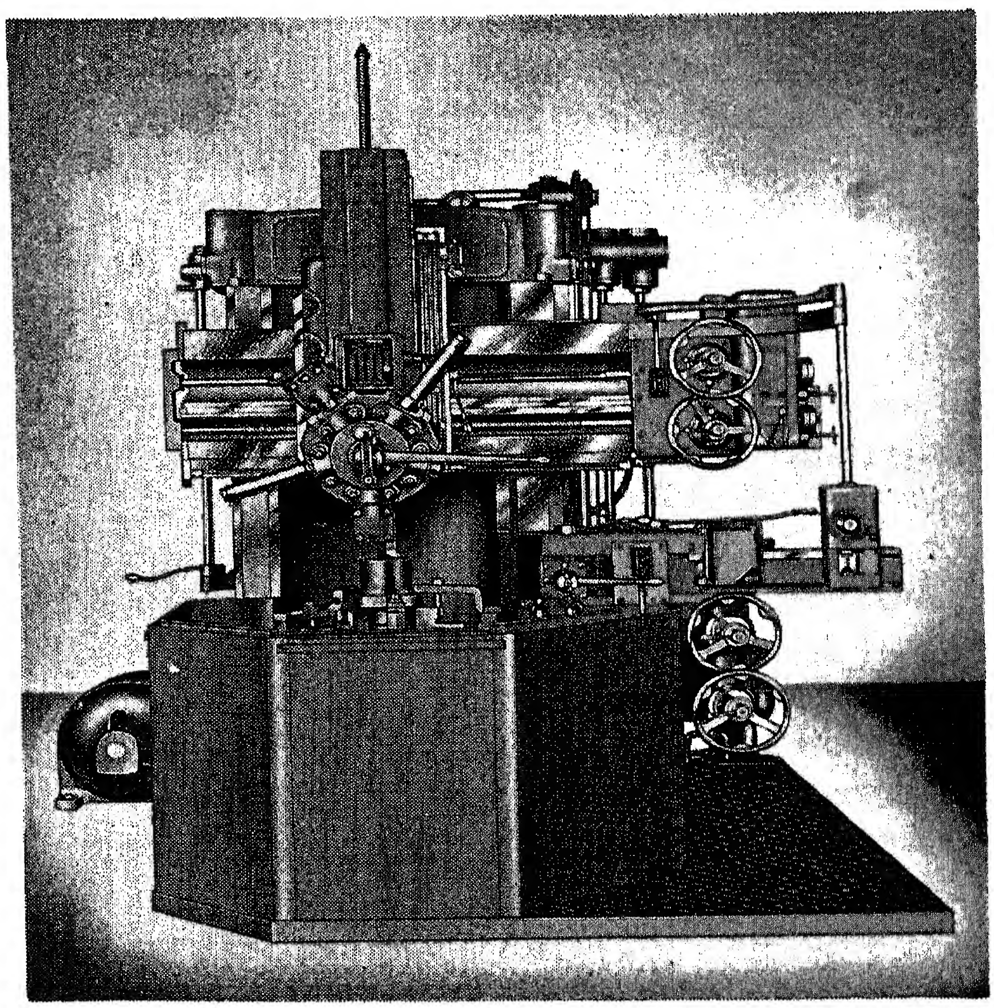


Fig. 14-3. A 36 in. vertical turret lathe. (Courtesy The Bullard Co.)

turret. The round work table revolves on heavy tapered roller bearings. Several kinds of table tops are available. A plain table has both parallel and radial T slots for stops and bolts for workpiece clamping. Tables are also made with built-in 4 jaw independent or 3 jaw combination chucks and radial slots. Sixteen table speeds from 5 to 140 rpm are provided on a typical 36 in. vertical turret lathe.

Just above and to the right of the table is a side head with a 4 station square turret. The side head slides on the column of the

machine. The square turret can be fed by power or hand, up or down, and away from or toward the center of the table.

Above the table is a counterweighted cross rail that can be raised or lowered and clamped in position on ways on the column. A saddle slides on the rail and carries a slide with a five-sided indexing turret. The saddle can be moved right or left by hand, positioned by an accurate leadscrew and dial or by a stop, and fed by power. The tools are fed to or away from the table by raising or lowering the turret and its slide. The turret can be positioned accurately by a screw and dial or fed by power. Sixteen feeds from 0.0025 to 0.500 in. per revolution are available for both cross and vertical movements on a typical 36 in. machine.

The turret is indexed and locked in each of its five positions by hand. All slides may be clamped for added rigidity when they need not be moved.

Tools in the turret head and side head may be applied to the work at the same time or at different times. Both multiple and combined cuts may be made on the vertical turret lathe, the same as on a horizontal turret lathe as described in Chapter 21.

The *Man-Au-Trol vertical turret lathe* is a type of vertical turret lathe that can be operated by hand but is also capable of going through a preset cutting cycle automatically. Each turret can be arranged to perform as many as 39 different and unrelated functions in any sequence without attention after being set up. Changes in spindle speeds and rates of feed of the tools and turret indexing are all coordinated with the cuts. With automatic operation, all the operator has to do is unload and load the workpieces. More time is required to set up the machine for automatic than for hand operation, but the difference is seldom more than a few hours, and automatic operation often is economical for lots as small as ten pieces.

Vertical boring mill. A vertical boring mill is like a vertical turret lathe but has no turret head, as illustrated in Fig. 14-4. It has a plain table with T slots for clamping bolts. The table rides on two flat circular tracks and is driven by a variable speed motor through a four speed transmission. A range of speeds from 0.6 to 19 rpm is available in small increments.

Normal equipment of a vertical boring mill includes two ram heads on the cross rail and one or two side heads. A ram head

consists of a saddle with a ram that can be fed up and down. The ram has a nonindexing toolpost on its lower end. Tools can be positioned vertically or crosswise by leadscrews and micrometer dials. The ram heads can be swiveled to incline the ram up to 60° either side of the vertical for tapers. Some side heads can also be swiveled in a vertical plane.

Vertical boring mills handle essentially round and symmetrical workpieces like reduction gear housings, turbine casings, and loco-

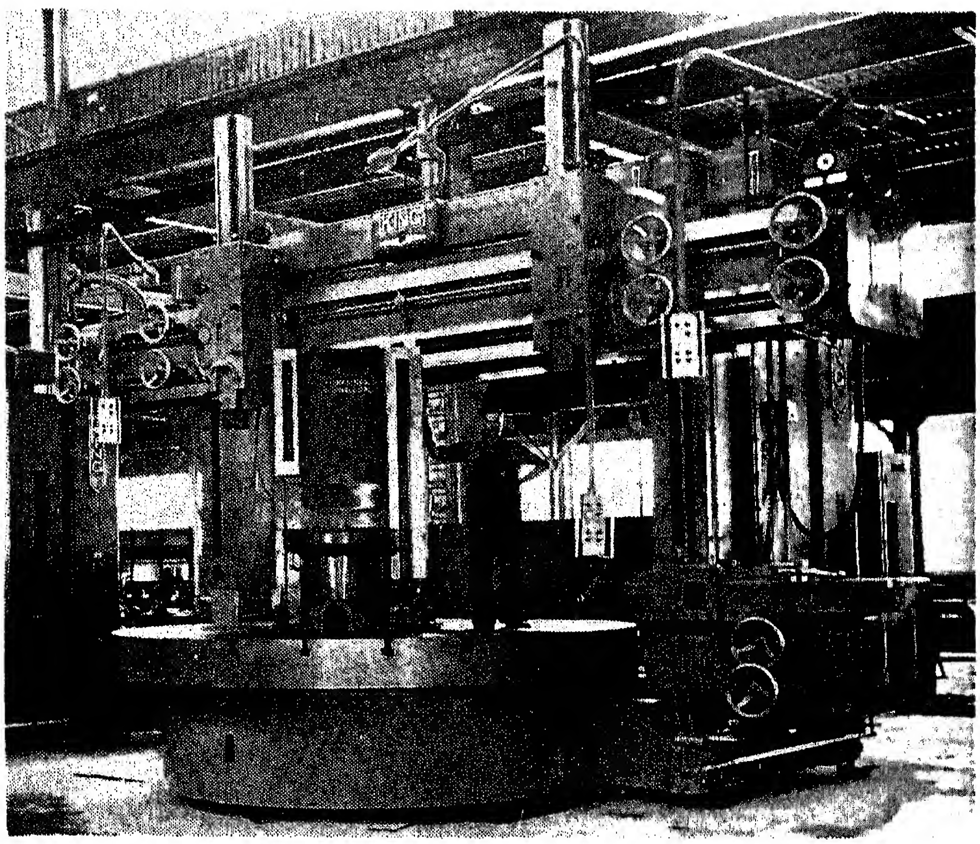


Fig. 14-4. A 144 in. vertical boring mill with the column of a 30 in. vertical turret lathe on its table. (Courtesy King Machine Tool Division, American Steel Foundries.)

motive tires. The size of the work is limited by the diameter of the table, the columns supporting the rail, and the height to which the tools can be raised. The machines do facing in a horizontal plane, boring, turning, grooving, etc. concentric with the axis of table rotation.

Vertical multiple spindle chucking machines. A vertical multiple spindle chucking machine has a number of stations at which work is done simultaneously for large quantity production. A typical one is shown in Fig. 14-5. It may have six or eight spindles.

each with a chuck or fixture on its upper end. The cutting tools are carried on vertical slides at evenly spaced stations around the central column of the machine. One station is for loading and unloading and has no tool slide. The chucks revolve at all but the loading station. The tool slides descend and feed the tools to the workpieces. When finished, the tools are retracted, and all chucks are indexed to succeeding stations. A finished piece is produced at each index in the time of the longest single operation plus a few seconds for indexing.

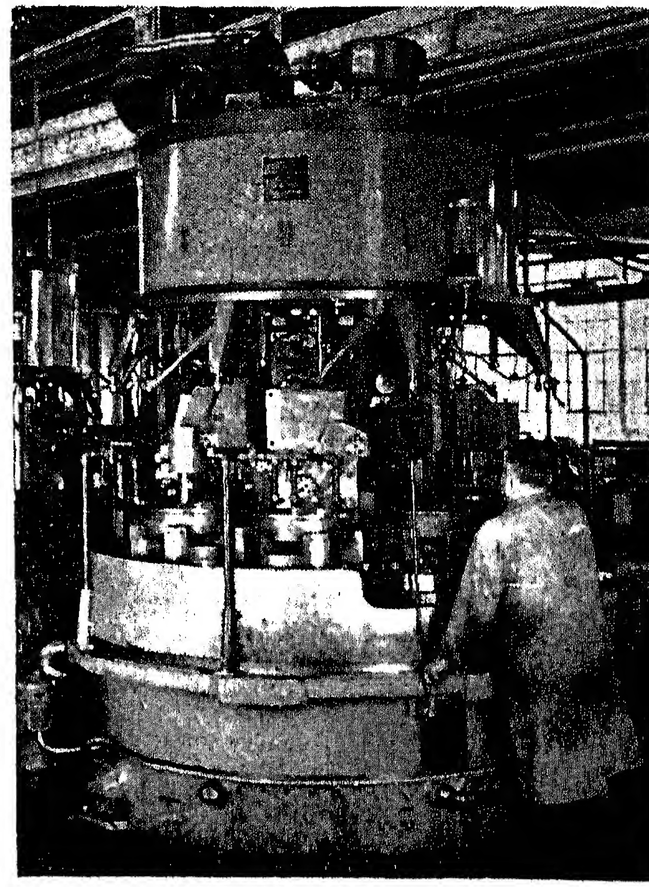


Fig. 14-5. A 16-23 in.-8 spindle vertical chucking machine. (Courtesy The Bullard Co.)

These machines are made both single and double indexing. On the latter, two pieces are loaded in chucks next to each other. Thus, a piece finished on both sides may be obtained at each index of the carrier. Typical machine sizes are 8, 12, 16 and 23 in., designating approximately the diameters of work handled. Drilling, boring, turning, facing, threading, or grooving can be done on forgings, castings, or cut-off bar stock.

Precision Boring Machines

Precision boring machines make use of single point tools to machine surfaces rapidly and precisely. They are capable consistently of holding tolerances of a few ten thousandths of an inch and finishing surfaces of 10 to 20 microinches rms or better. As their name implies, they are used mostly for boring, but also are arranged for facing, turning, grooving, and chamfering. The work they do can also be done on general-purpose machines, such as the lathe, but the precision boring machines are more efficient where

large quantities of parts are produced because they operate in semi-automatic cycles. The machines themselves are reduced to essentials and are relatively simple and inexpensive, but special tooling must be provided for almost every job they do.

Most precision boring machines have horizontal spindles, but on some the spindles are in vertical or angular positions for particular jobs. As an example, spindles are arranged in two inclined banks on a machine for boring all the cylinders at one time in a V-type engine block. As a rule, the spindles revolve the cutting tools, but sometimes they carry fixtures and revolve the work.

The work is fed to the tools on some machines, like the one in Fig. 14-6. On others the spindles are fed to the work, as in Fig. 14-7. One class of precision boring machines is fairly flexible and can be adapted to many jobs with suitable tooling. They are looked upon as standard machines. Others are made for special applications and are not readily changed to other purposes.

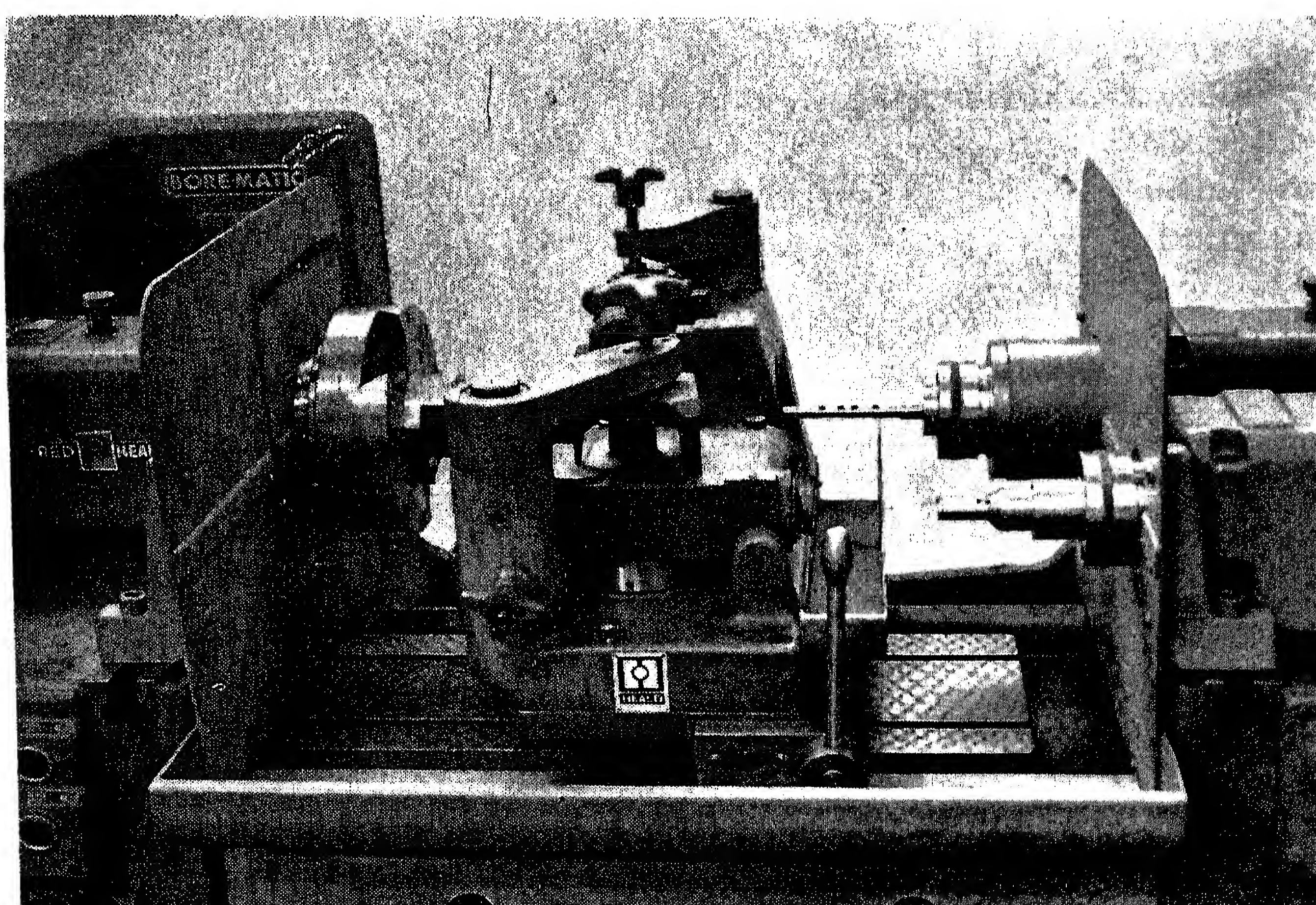


Fig. 14-6. A view of the work station of a double end horizontal precision production boring machine with tooling for machining 10 surfaces on a pump body. (Courtesy The Heald Machine Co.)

Standard precision boring machines. A popular type of precision boring machine made in various standard sizes has a table that reciprocates longitudinally on a bed. The table is driven hydraulically and can be made to traverse at any rate in a wide range, dwell, or reverse at any point in its stroke. The action is governed by dogs on the side of the table. They are set to trip hydraulic valves. At one or both ends, a bridge attached to the bed spans the table and carries one or more boring heads.

A typical double end precision boring machine of basic standard construction tooled for production of a pump body is illustrated in Fig. 14-6. This is called a double end machine because it has boring heads on bridges at both ends of the table. A special two-station fixture is bolted to the table and locates the part on its side in the front station and in an upright position in the rear station. When the preset operation cycle is started, the table moves rapidly to the left and then proceeds at feed rate while the head on the left-hand bridge bores a 3.999-4.000 in. diameter in the part in the rear station. Upon completion of the boring, the table is stopped, and the cross-feed head on the boring spindle comes into operation and machines two faces. The facing is done by tools held by a toolholder that feeds outward on the revolving cross-feed head. At the conclusion of those cuts, the table traverses to the right. The heads on the right hand bridge bore and face both pieces. When that is finished, the table returns rapidly to its original position and stops.

The job just described is one of many for which machines of this type can be aranged and tooled. The heads can be adjusted to desired positions on the bridges, and double spindle heads are available for close center distances. At least a moderate amount of production is necessary to justify the cost of the special tooling required for each job.

A variation of this kind of machine is one on which the heads are arrayed alongside the table with their spindles at right angles to the direction of table movement. Such machines are used for facing and straight grooving.

Tools for precision boring machines. Diamond and cemented carbide tipped tool bits are commonly used on precision boring machines because they cut rapidly, hold sizes, and produce good finishes. Various types of toolholders are mounted on the tables when the work is revolved. They include plain, rigid cutter blocks,

and manual or automatic eccentric tool retracting units that withdraw the bits from the surfaces at the ends of cuts to avoid backtracking marks.

Rotating toolholders or boring bars used on precision boring machines are called quills and usually are made with a flange at one end that is bolted to the end of a boring head spindle. An eccentric pilot on the quill flange fits into an eccentric hole in the end of the spindle. Thus as the quill is turned to different positions on the spindle, it is moved off center different amounts. That varies the diameter to which a tool bit cuts and serves as an adjustment for the size of cut. The bit or bits are held in slots or holes in the bar

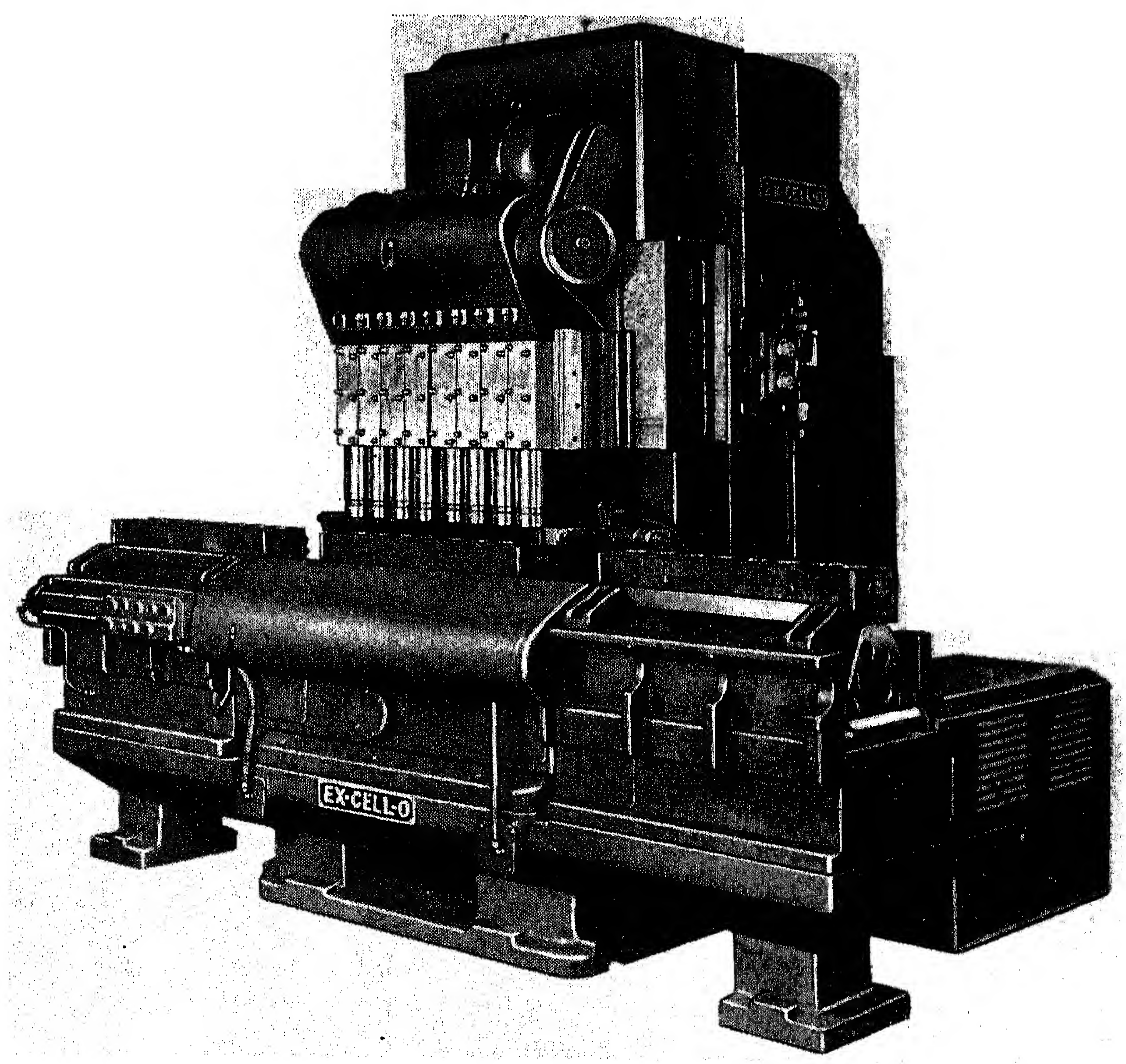


Fig. 14-7. A special precision production boring machine for boring all the cylinders in an engine block at one time. (Courtesy Ex-Cell-O Corp.)

that protrudes axially from the quill flange. Boring quills generally are made to suit each specific job.

Various types of universal fixtures are available to improve the versatility of standard precision boring machines. A common type consists of an angle plate with T slots and tapped holes on its face for mounting supplementary fixtures and toolholders. The plate can be adjusted in three coordinate directions on a base. Indexing and rotary fixtures and bases also are often used to position work-holding fixtures and workpieces.

Special precision boring machines. Entirely special precision boring machines are constructed for particular purposes where the output is sufficient to justify the cost. A machine of that kind that bores all the cylinders at once in an engine block is shown in Fig. 14-7.

Jig Boring Machines

Precision hole location. Methods of locating and machining holes on the lathe, drill press, and milling machine have been discussed in Chapters 6, 11, and 13. The accuracy required for the location of holes varies from 1/32 in. commonly needed for clearance holes to a few ten thousandths of an inch for holes in exacting production parts, such as those for aircraft engines, and many jigs, fixtures, dies, gages, and other tools. The methods that have been described are frequently not adequate nor feasible when holes must be positioned within a few thousandths or ten thousandths of an inch. Other methods must be employed where higher degrees of accuracy are required, and those methods will now be presented.

The accurate location of holes is an especially important consideration in toolmaking. In the first place, jigs, fixtures, dies, gages, and other tools must be much more accurate than the parts they help produce. Holes often establish points of reference or location in tools. For instance, indexing in a jig or fixture is commonly done by locating in accurately spaced holes. Pins and plugs that locate workpieces on jigs and fixtures usually are positioned in holes. The positions of the holes in cutting and forming dies determine the accuracy of the pieces produced. Tools are generally made in small quantities, and require general-purpose equipment for their fabrication.

The accurate location of holes calls for three steps. The first is to establish the positions of the holes, the second is to cut the holes, and the third is to check the results.

The positions of holes may be established by layout, buttoning, transfer, and coordinate location.

The location of holes by layout and its limitations were described in Chapter 11. Layout is not a precision method.

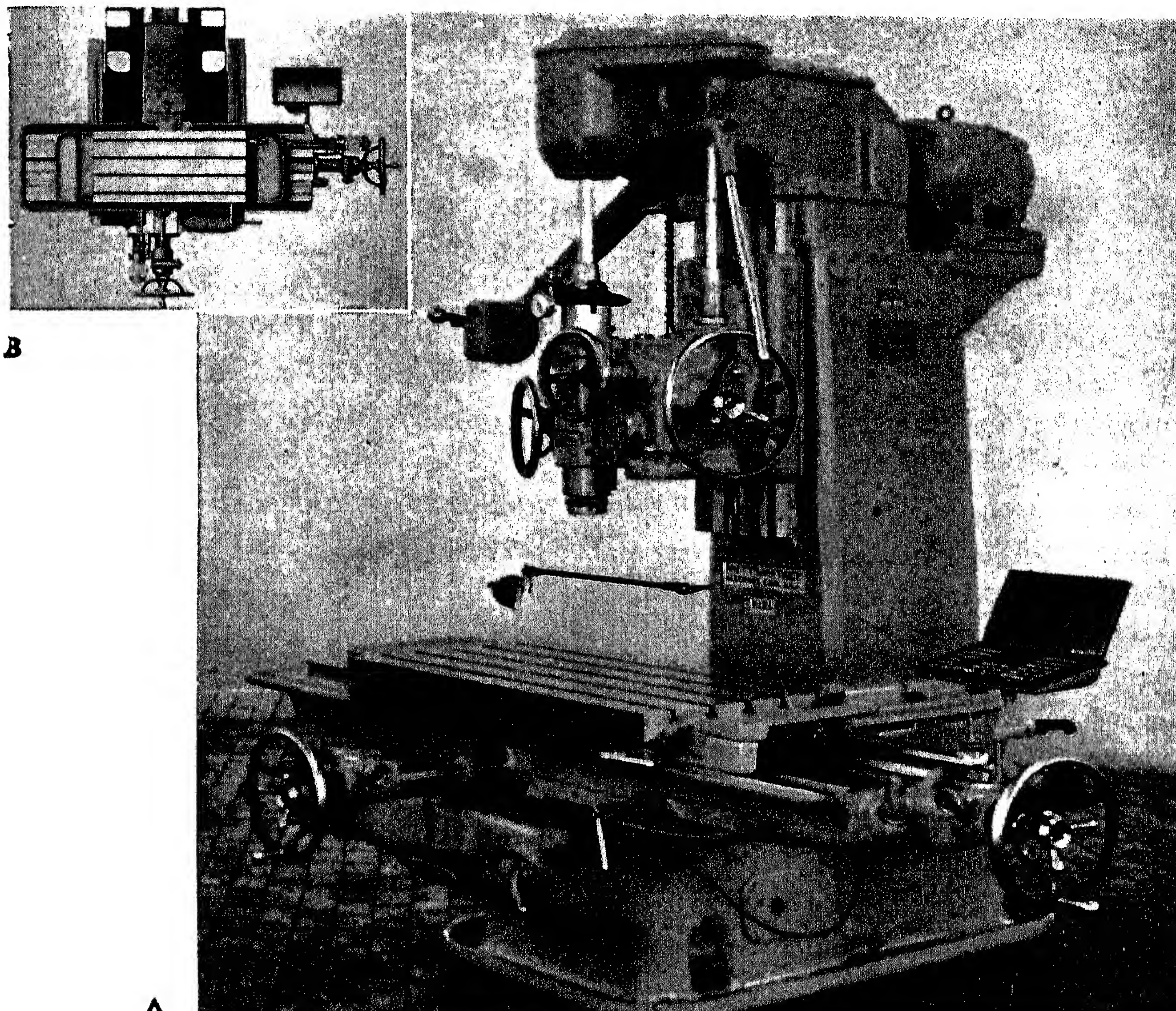
Buttoning is the name of a method using *toolmakers' buttons.* These are accurately sized hollow cylinders with squared ends. A button is clamped by a screw to the workpiece and adjusted with precision measuring tools to the position desired for a hole. The workpiece is then mounted on the face plate of a lathe with the button protruding. The workpiece is shifted until a dial indicator shows the button running true. The button is then removed, the workpiece secured in place, and the hole is bored in the same spot. Holes may be located by this method to within 0.0005 to 0.001 in. of true location. No expensive equipment is needed, but the method is time consuming.

The transfer method of hole location involves the use of templates or jigs. Jigs are adequate for producing the accuracy of hole location required in almost all parts produced in quantity, but generally cannot be justified where one or a few pieces are made.

Coordinate location is accomplished by moving a workpiece from a reference point through accurately measured distances in coordinate directions normal to the cutter spindle. This can be done repetitiously with the hole spacer illustrated in Fig. 11-18 and described in Chapter 11. Holes can be located along coordinate lines on such general-purpose machines as the milling machine and horizontal boring mill by setting off the required distances by means of the leadscrews and graduated dials on the machines. On such machines, holes can be located within 0.001 to 0.005 in. of their true positions without extra attachments. Coordinate location is also done on jig boring machines, but the means of measurement are refined, and location may be achieved within 0.0001 to 0.0005 in. if necessary.

After the position of a hole has been established, the way it is cut determines whether accurate location as well as size ensues. Where a truly positioned and round hole with an accuracy of less than half a thousandth in. is required, drilling followed by several cuts with

a single point boring tool is the only means of insuring it. Faster methods may be used if more tolerance is permitted. For tolerances between 0.0005 and 0.001 in., a multiple point boring tool or end mill followed by a sizing reamer is recommended. Where tolerances



B

A

Fig. 14-8. A jig boring machine that employs measuring instruments. (Pratt and Whitney Photo from Pratt and Whitney Division, Niles-Bement-Pond Co., W. Hartford, Conn.)

of several thousandths of an inch are permissible, a hole may be spotted, drilled undersize, and reamed.

Holes are finished in hardened materials by lapping, honing, and grinding. Accurate diameters and good surface finishes can be produced by these methods. Honing and lapping are not effective in altering the position of a hole, but grinding is a positive way of correcting positional errors.

The positions of holes may be measured directly with precision measuring instruments. Jig boring machines are frequently used to check hole positions.

Jig boring machines. A jig boring machine looks like a vertical milling machine. Most jig borers in the United States are of the openside construction like the one in Fig. 14-8. The vertical spindle moves up and down with its quill in a bracket clamped to the front of the column and adjustable for height. A stop or depth indicator is provided for limiting the depth of travel of the spindle as desired. The table is mounted on a saddle on the bed and can be moved lengthwise or crosswise. Large pieces on the table can extend out on the sides or front, and the operator has ready access to the work.

Another style of jig borer has its table sliding lengthwise directly on the bed. Two columns, one at each side in the middle of the bed, support a cross rail that carries the vertical spindle head. The head is moved across the table, and the spindle is fed vertically.

Jig boring machines are characterized by provisions to insure accuracy through rigidity, low thermal expansion, and precise means of measuring distances. The spindles of most machines run in preloaded antifriction bearings. The spindle housing may be cast of Invar iron with a low coefficient of expansion. A wide range of speeds is available so that cuts may be taken quickly. Measurements are made and workpieces positioned in different ways on different makes of machines. The four basic measuring devices on jig boring machines are micrometer leadscrews, graduated scales and microscopes, end measuring instruments, and the Electrolimit measuring system.

All jig borers have leadscrews and graduated dials for moving their tables, but the screws are not always depended upon for precise measurements and table settings. Accurate screws are difficult to make, particularly long ones, and are subject to wear. Screws that move table and saddle must exert sizable forces that cause deflections. The use of screws alone has been found satisfactory for small and medium-size jig boring machines. The forces exerted need not be large. Short screws can be cut and hardened with an error of no more than 0.0002 in. in 16 inches.

If long screws are used, corrections must be made by a compensating device. This is done on some Swiss jig boring machines in the following way. The error in table movements on the machine is

carefully measured, and a strip cam is made with a profile curve representing the errors to a magnified scale. The cam is fitted to the side of the table, and a lever follows the cam profile as the table moves. The variations in the cam are transmitted by a linkage to the vernier on the leadscrew dial. The vernier's position is shifted so that it points to true readings on the dial. The same kind of arrangement corrects the cross movement readings. Since the leadscrews serve to move the table as well as to provide the measurements, machines of this type can be manipulated rapidly.

Measurements are made on some jig borers by means of accurate vernier scales that are read by microscopes.

The jig boring machine of Fig. 14-8 is equipped with end measuring instruments. The end measures are rods of even inch lengths made to gage block accuracy. An inside micrometer is adjusted for decimal parts of an inch. The end measures and micrometer are placed in a trough between an adjustable stop on the table and a 0.0001 in. dial indicator at the outer end of the trough. The table is locked in starting position, measuring instruments are inserted, and the table stop is adjusted to set the dial indicator to zero. A hole may be bored in that position. To locate the next hole, the measuring instruments are changed an amount equal to the dimensional difference desired, and the table is moved until the dial indicator again registers zero. The same procedure is followed to set the saddle for cross movements.

The effects of deflection and wear are minimized by the end measuring system. Wear in the screw has no effect upon the measurements. The measuring device is subject only to a slight uniform pressure from the indicator which helps keep the rods together.

A jig boring machine equipped with the electrolimit measuring system has a bar with a series of projections attached to the side of the table and a like bar on the saddle. The projections on the bar are magnetized, and the magnetic centers of adjacent projections are 1 in. apart within a claimed accumulated tolerance of 0.00002 in. in the full length of the bar. The bar is carried past an electromagnetic head when the table is moved. A meter shows when the head and a projection are exactly in line. For measurements of less than 1 in., the head is adjusted by a precision micrometer screw and dial within 0.0001 inch.

Jig boring tools and operations. Jig boring machines are used

not only for accurate toolmaking, but also to manufacture parts in small quantities and for close limit inspection. Although the jig borer is thought of mainly as a machine for producing holes, it often is put to milling surfaces and grooves and even some turning and hollow milling. A jig boring machine and its accessories represent a large investment but provide means for locating holes and doing other precise work more accurately than can be done on most other machines and more rapidly than can be done by layout or buttoning.

A variety of work is usually done on a jig boring machine, and much more time is consumed for setup and changeover than for cutting. Thus, appreciable saving in time can be realized from the tools and accessories that make manipulation of the machine quick and easy.

Positioning the workpiece on the table with respect to the spindle of the machine is the first major step in a jig boring operation. Often holes must be bored at specified distances from two square sides of a workpiece. The sides must be aligned parallel with the movements of table and saddle. That is done by means of an indicator attached to the spindle. The stem of the indicator is brought against a side, and the workpiece is shifted until the indicator shows no change as it traverses the surface.

An indicator is mounted on the spindle to center the spindle with an edge of a workpiece, a hole, or a pin. A gage block is placed against the vertical face to "pick up" an edge. The indicator is read with its stem against the work face, the spindle is turned 180°, and a reading is taken from the side of the block in contact with the work face. Adjustments are made until the two readings are the same.

For a hole or pin, the spindle is turned slowly with its attached indicator touching the surface to be centered with the spindle. Adjustments are made in the position of the work until the indicator pointer stands still all around the surface. Standard brackets and adjustable *grasshopper legs* are available for attaching indicators to jig boring spindles. The same procedures for "picking up" edges, holes, etc. are followed when inspecting work on the jig borer.

An inclined microscope with crosslines that coincide with the center of the spindle may be attached to the end of the spindle. With that, the center of the spindle may be located accurately over the edge of a piece or over scribed lines. A true proving bar of accurate diameter may be mounted on the spindle. The spindle is

positioned so that a gage block just fits between the bar and the workpiece. The center of the spindle is then a known distance from the vertical work surface.

Auxiliary tables that tilt and rotate are useful for doing jobs that would otherwise be difficult or impossible. For instance, such a table saves considerable time in locating holes where compound angles are involved. Some tables are driven only by hand, others by

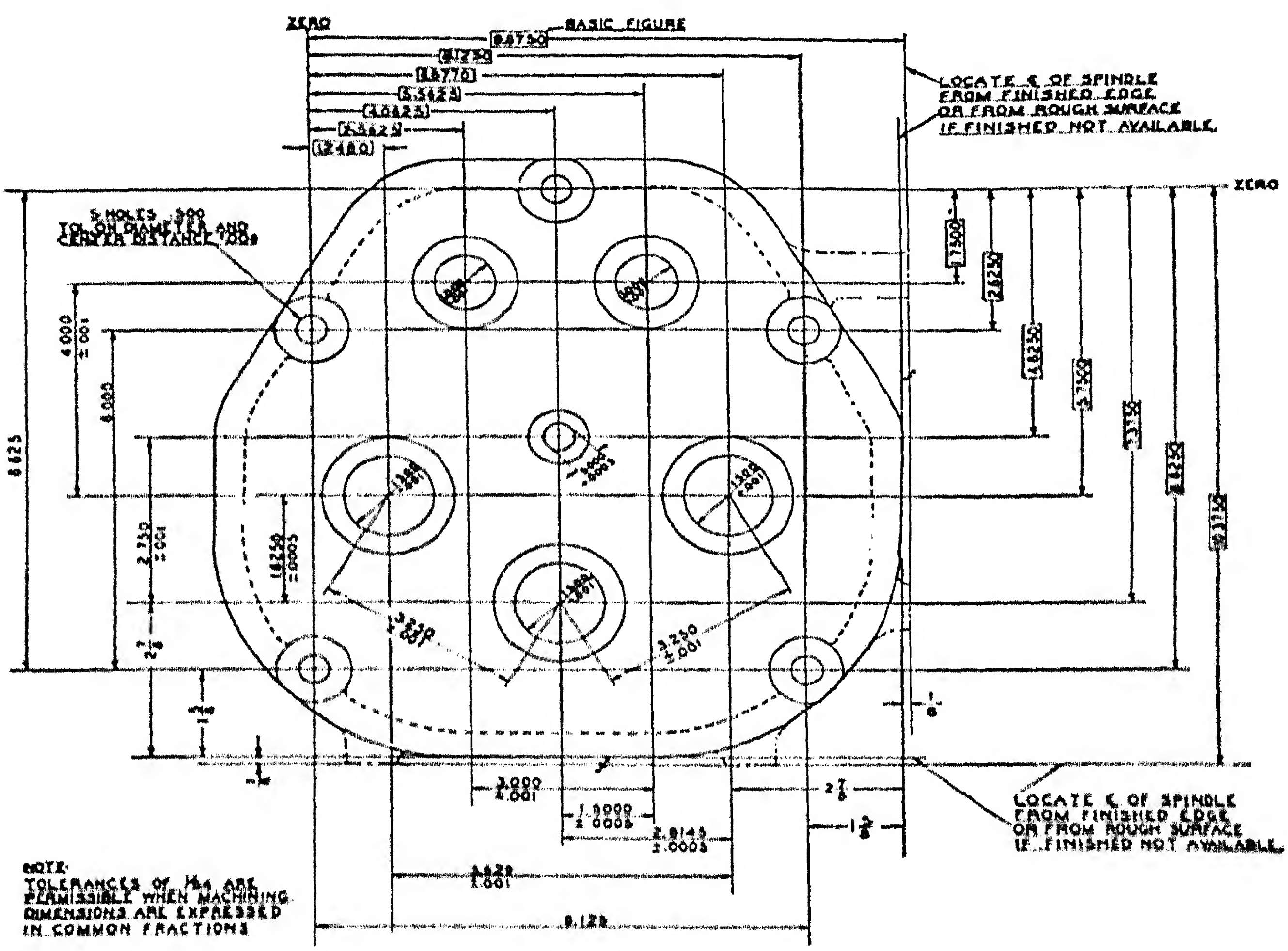


Fig. 14-9. An example of the way to dimension a part for jig boring. (Pratt and Whitney Photo from Pratt and Whitney Division, Niles-Bement-Pond Co., W. Hartford, Conn.)

hand and power. Tables 12 to 30 in. in diameter are available for American jig boring machines. They have micrometer dials graduated in degrees, minutes, and seconds and are guaranteed accurate to 0.00025 in. on a 10 in. radius.

Workpieces are most commonly held with bolts and strap clamps. A large variety of parallel bars and precision angle irons are also used. Center punches and scribing tools are mounted in the spindle

for layout work. The cutting tools employed include center drills, drills, end cutting end mills, boring bars, single point boring tools, and reamers in a large variety of sizes. The cutting tools may be held in drill chucks, collets, boring heads, etc. or inserted directly into the spindle. Generally adjustments in workpiece position can be made more easily than the tools can be changed on a jig borer. Also changing of tools adds a certain amount of error. Consequently a desirable procedure is first to make all roughing cuts and then the finishing cuts on all holes in a workpiece.

Jig boring is done most efficiently if all points can be referred to two coordinate axes that can be established at the outset of the operation. An example of a drawing dimensioned to carry out this principle is given in Fig. 14-9. Coordinate basic dimensions enclosed by rectangles in the drawing are given from an axis at the top and one at the left. These are the dimensions the operator uses to set the measuring instruments, and they permit him to move from hole to hole merely by inserting the correct end measures and setting the inside micrometer. He does not have to do any figuring but can work directly to the basic figures on the drawing. The dimensions not enclosed in rectangles show the required tolerances. They indicate what methods should be used to finish the holes and guide the inspector in checking the finished part.

Temperature changes are important in working to small tolerances on a jig borer. Iron and steel expand about 0.000006 in. per degree Fahrenheit rise in temperature for each inch of length. A 10° F change is barely perceptible but causes a difference in size of 0.0006 in. in a 10 in. dimension in a steel part, and that in itself can impair precision work. Temperature changes of at least 10° F may result from the handling of measuring instruments, the action of the cutting tools, or heat from motors and pumps on the machine. Precision work requires that the room temperature be maintained at 68° F. In any event, care and time must be taken to allow heat to be dissipated from the machine and work for accurate jig boring.

Jig grinders. A jig grinder is like a jig borer except that the spindle of the machine carries a high speed grinding spindle that revolves in a planetary fashion. The jig grinder is always used for finishing operations, generally on hardened steel, and must be built to the highest degree of precision and be designed to maintain that precision. Jig grinders are capable of finishing holes in hard

materials to a degree of accuracy equal to that of jig boring in soft materials.

Questions

1. What are the standard types of horizontal boring mills? For what purposes are they used?
2. Describe a table-type horizontal boring mill. How is its size designated?
3. What tools are commonly used on horizontal boring mills?
4. How does a vertical boring and turning machine resemble a lathe? For what kinds of work is the vertical boring and turning machine advantageous?
5. What distinguishes vertical turret lathes from vertical boring mills?
6. How may the work be held on a vertical turret lathe? On a vertical boring mill?
7. Describe a vertical multiple spindle chucking machine. To what work is it applicable?
8. What are the advantages and disadvantages of precision boring machines?
9. What kinds of tools are used on precision boring machines?
10. Name and describe briefly four methods for locating holes. What are their relative advantages and disadvantages?
11. What four basic devices are found on jig boring machines for making measurements and positioning workpieces?
12. Why are leadscrews alone not desirable for measuring and positioning? When may they be used?
13. Describe the use of end measuring instruments on jig borers.
14. How may a workpiece location be "picked up" on a jig boring machine?
15. How should a drawing be dimensioned to facilitate jig boring?
16. Why are temperature changes important in precision jig boring? What precautions must be taken to control temperature?

References

Handbook for Horizontal Boring, Drilling, and Milling Machines. Fond du Lac, Wis.: Giddings and Lewis Machine Tool Co., 1947.

Manufacturers' Catalogs:

Bullard Co., The, Bridgeport, Conn.

Excello Corp., Detroit, Mich.

Giddings and Lewis Machine Tool Co., Fond du Lac, Wis.

Heald Machine Co., The, Worcester, Mass.

King Machine Tool Division, American Steel Foundries, Cincinnati, Ohio.

Lucas Machine Division, The New Britain Machine Co., Cleveland, Ohio.

Pratt and Whitney Division, Niles-Bement-Pond Co., W. Hartford, Conn.

Chapter 15

SAWING AND FILING

METAL IS REMOVED IN SAWING AND FILING by the action of many small teeth. Saw teeth act in a narrow line, and a saw can sever a sizable chunk of metal with a minimum amount of cutting. If a piece of metal is removed by milling, for instance, a large part or all of it may have to be reduced to chips. The same piece may be cut off or out by a saw acting on only a small part of the material. Thus, in many cases work can be done faster and with less power by sawing than by other methods of metal cutting, and material can be saved.

The teeth of a file act over a wide surface and progress slowly. Their cutting effect can be discerned readily and controlled. Thus, filing is suited for finishing irregular surfaces and surfaces difficult to reach with other kinds of cutting tools. Filing is limited to removing small amounts of soft materials.

Because they can be applied slowly, with little force and power, saws and files have been used as hand tools since ancient times. But they also are power driven at greater rates. Power-driven hack saws, circular saws, and band saws are employed to cut off pieces of bar stock, plates, sheets, and other shapes of metal. Versatile band saw machines have been developed for cutting out dies, punches, details of jigs, fixtures, gages, and even production parts with a minimum waste of material and time. Reciprocating and continuous filing machines are available for rapid and accurate finishing of irregular surfaces in small quantities.

Sawing Machines

Power hack saw machines. A power-driven hack saw machine drives a blade back and forth through a workpiece, as a person does

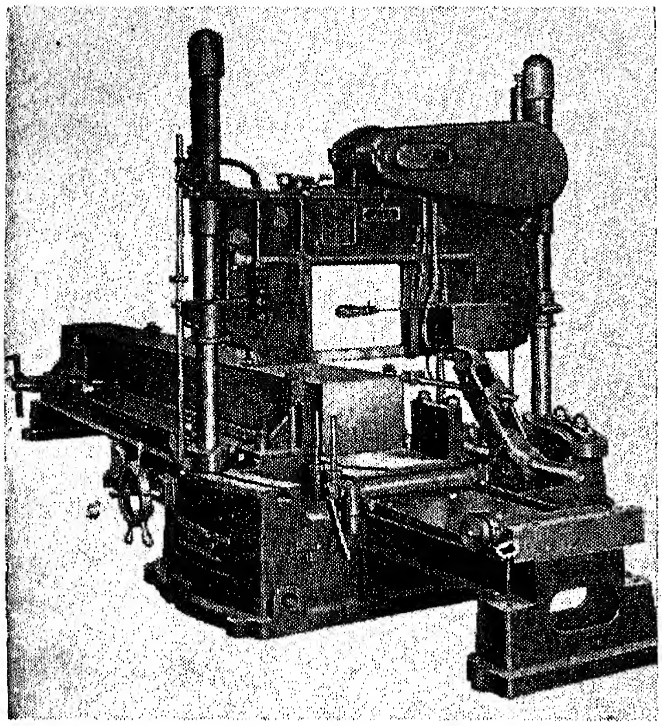


Fig. 15-1. An 18 in. by 18 in. capacity power hack saw with an 18 in. by 18 in. by 12 ft long billet on the machine. Only one size larger than this hack saw is made. (Courtesy Armstrong-Blum Mfg. Co.)

in using a hand hack saw. The saw is pressed against the work on the cutting stroke but raised a little to clear the work on the return stroke so the teeth do not drag. Small and medium-size machines are crank driven; large machines are hydraulically driven. A quick return motion on the back stroke saves time. Pressure to feed the saw may be applied by the weight of the saw frame, extra weights, springs, a positive screw feed, or hydraulic pressure. Under gravity or spring tension, the feed is regulated by a ratchet mechanism. Cutting pressures can be controlled accurately with hydraulic feed, which is advantageous for light work like thin walled tubing.

A vise is the usual means of holding the work on a power hack saw. It is fixed to the base of the machine. Commonly a coolant reservoir, pump, and supply line are part of the machine equipment. An automatic cut-off stops the machine at the end of each operation. Hack saw machines driven by fractional horsepower motors and capable of cutting stock up to several inches square are applicable to general-purpose work where speed is not important. Heavy machines of several horsepower capacity are used for large pieces and for production. One of these that takes stock up to 18 in. square is shown in Fig. 15-1. Its saw frame can be swiveled for angular cuts. Production machines of this type can be arranged to feed, measure, and cut off a series of pieces automatically from one or more bars.

Because the stroke is intermittent, hack sawing is not a rapid method of cutting off stock, but the machines are simple in design, flexible, and easy to operate and maintain. The blades are relatively inexpensive.

Circular saw machines. A circular saw machine cuts off stock

with a rotating saw. The commonest type utilizes a cold saw that has teeth and operates like a thin milling cutter. The saw is mounted on a carriage and is fed through the work in the manner indicated in Fig. 15-2. The feed is manual on some machines but usually is air or hydraulic driven and can be varied to get the most out of the saw without harming it. The saw speed can be changed by pick-off gears to get a suitable rate of speed for the material cut.

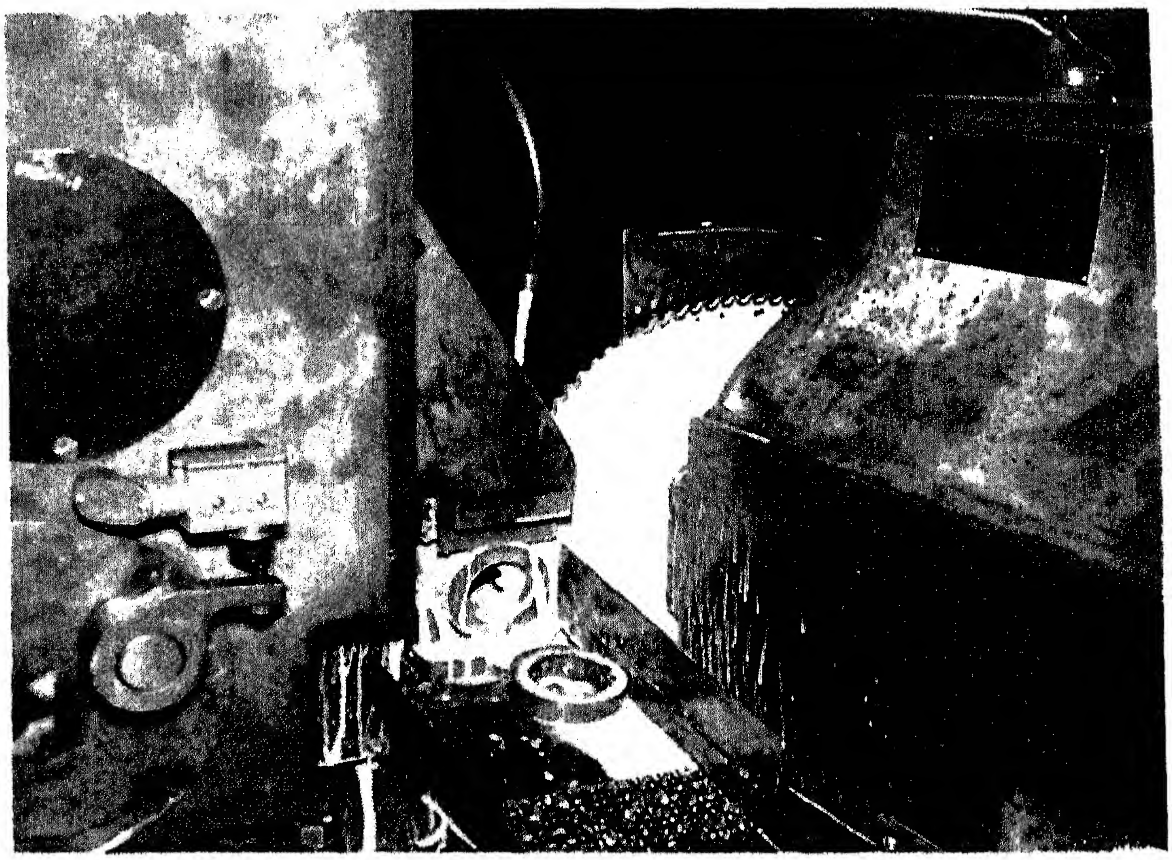


Fig. 15-2. A circular sawing machine that uses a cold saw to cut stock up to 4 in. square or round. (Courtesy The Motch and Merryweather Machinery Co.)

The work is usually clamped in a vise or V fixture after being pushed against a stop to gage the cut-off length. Generally one piece is cut at a time, although more may be cut when advantageous to do so. Large machines for heavy work may have rollers to aid in loading the pieces.

Manual, semiautomatic, and automatic circular sawing machines are available. On the manual models, the stock is positioned and clamped and the saw is fed by hand. On the semiautomatic machine, the operator feeds a bar each time against a stop. One or more of the subsequent functions may then occur automatically.

An automatic machine feeds a bar to a stop, clamps it, cuts off a piece, and then repeats the cycle until the bar is all gone. One man can attend to several such machines because he has only to load the bars. A high production model chamfers or centers the pieces after they are cut off.

Cold sawing is a continuous and fast method for cutting off and is preferred over other methods for many production jobs, especially

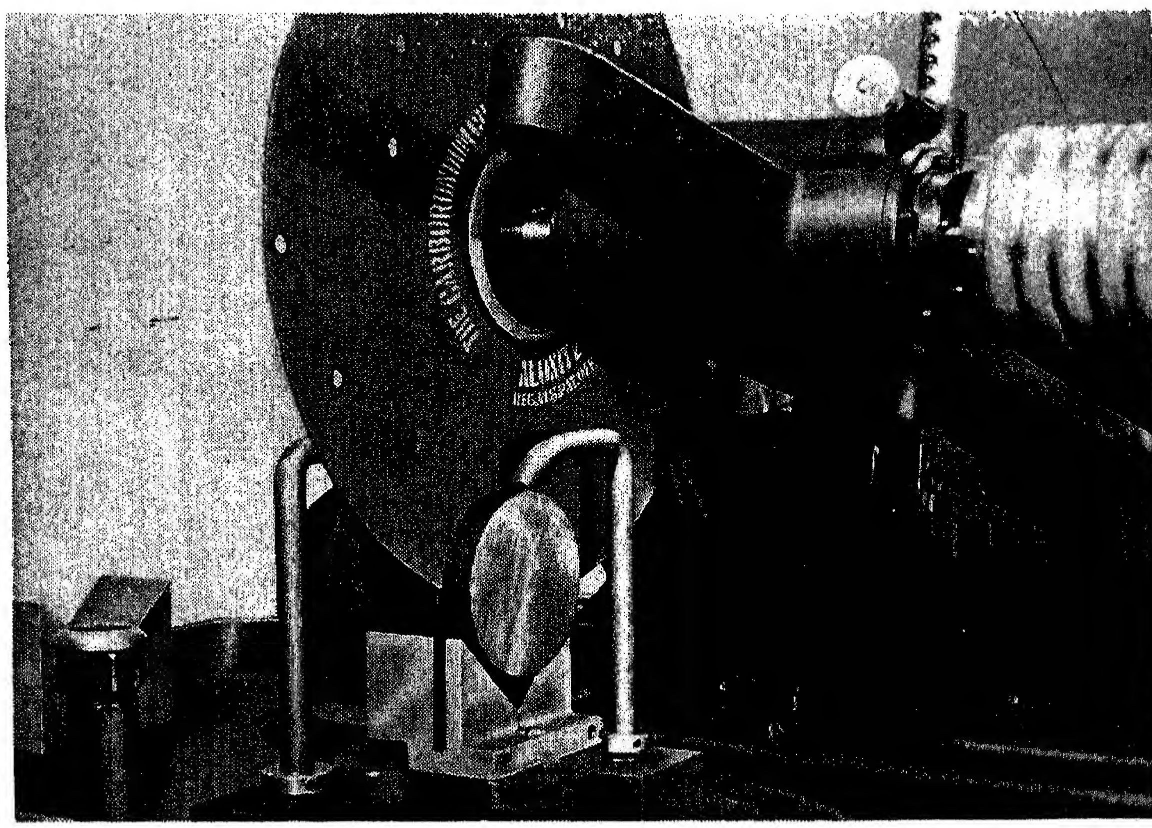


Fig. 15-3. An abrasive cut off wheel mounted on a high speed swing frame machine. The holes in the wheel help carry coolant into the cut. (Courtesy The Carborundum Co.)

for cutting off bar stock and other shapes in preparation for subsequent operations. A 6 in. diameter steel bar can easily be cut off in a minute. The cut-off faces are left with a smooth milled surface and few or no burrs. The ends can be made square and accurate. The typical experience of one plant is that pieces of bar stock can be cut off with a tolerance in length of 0.003 in. and a finish equal to or better than that obtained from a cut-off tool on a lathe. Cold sawing is done on pieces up to 16 in. diameter.

Another kind of circular saw is the friction saw. It has a smooth or nicked outer edge and is run at 10,000 to 25,000 sfpm. Diameters from 2 to 6 feet are common. The heat of friction softens the metal

in contact with the disk, and the soft metal is rubbed away. Only a small amount of the saw is in contact at any instant, and the rest is cooled as it travels around to enter the cut again.

Abrasive disks, like the one shown in Fig. 15-3, also are used for cutting off stock. As the name implies, an abrasive disk is a thin flexible grinding wheel. It runs at speeds as high as 15,000 sfpm and is usually carried on a swinging frame.

Band saw machines. A continuous saw blade or band runs over the rims of two wheels on a band saw machine. Horizontal band saw machines, like the one in Fig. 15-4, are used for cut-off operations. The saw is carried on a frame and is fed downward under controlled hydraulic pressure. The work is held in a quick acting vise. Cutting fluid supplied by a pump is carried into the cut by the saw. After the cut has been completed, the saw is raised clear of the work.

For cutting off, band saws are continuous and faster but more expensive than hack saws. They remove a minimum of material, and the depth of cut can be controlled when the blade travels parallel to the bed of the machine. Band saws are not as fast

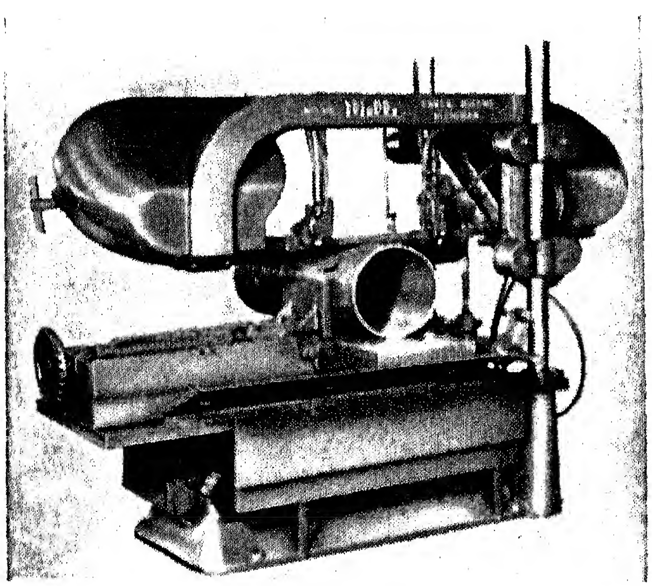


Fig. 15-4. A No. 12 horizontal metal cutting band saw with a 12 in. by 16 in. stock capacity. (Courtesy Wells Manufacturing Corp.)

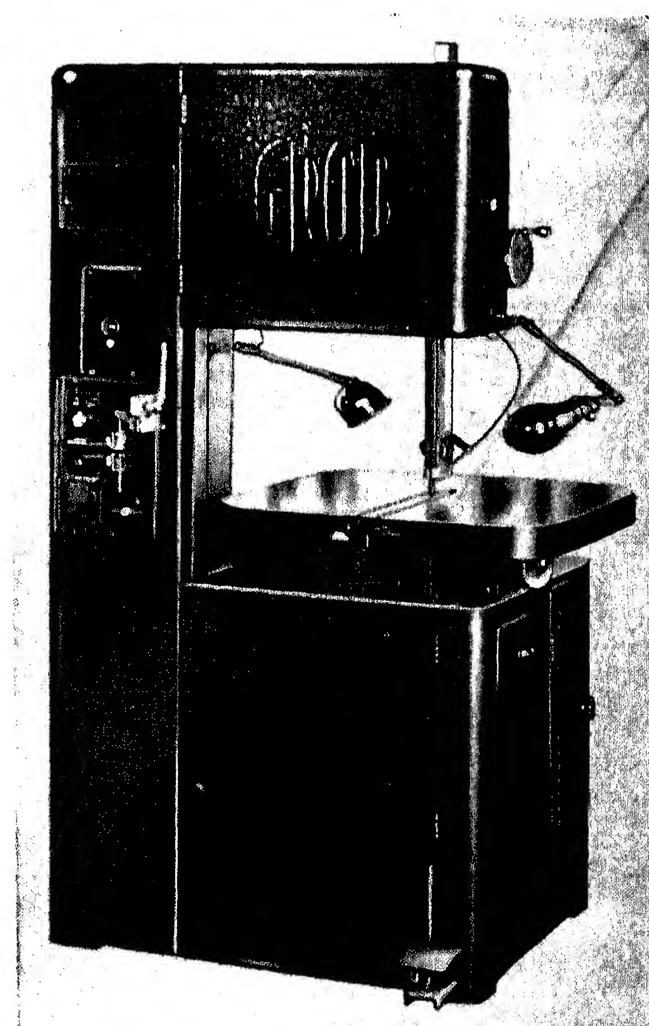


Fig. 15-5. A vertical contour sawing band saw with a 24 in. by 24 in. tilting table. (Courtesy Grob Bros.)

in many cases as circular saws but are preferred for some work. Low pressures can be applied to thin and fragile workpieces, which need not then be heavily clamped. Some castings and other irregular pieces hard to clamp can be held and fed by hand.

Vertical band saw machines are used to cut off objects of many shapes and all kinds of materials, metallic and nonmetallic. In addition, vertical band saws have been developed highly in recent years for precision contour sawing. In that role, these machines are capable of doing fast, efficient, and accurate work such as cutting out die openings, irregular jig, fixture, and gage details, and small intricate production parts.

A typical medium-size contour sawing vertical band sawing machine is shown in Fig. 15-5. The band saw travels over enclosed wheels above and below the table and is confined by adjustable guides on entering and leaving the work. Surface speeds of 50 to 2030 sfpm are available to handle a variety of materials. Speeds on some machines are as high as 15,000 sfpm. Band saw machines can be set up quickly and easily for many jobs. Much work can be placed on the table and

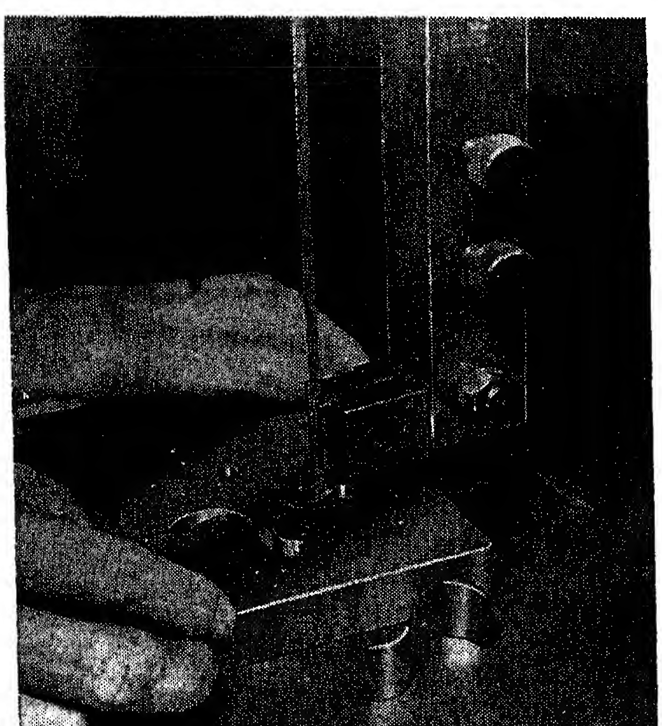


Fig. 15-6. Openings in a die block being sawed on a contour band sawing machine. (Courtesy Grob Bros.)

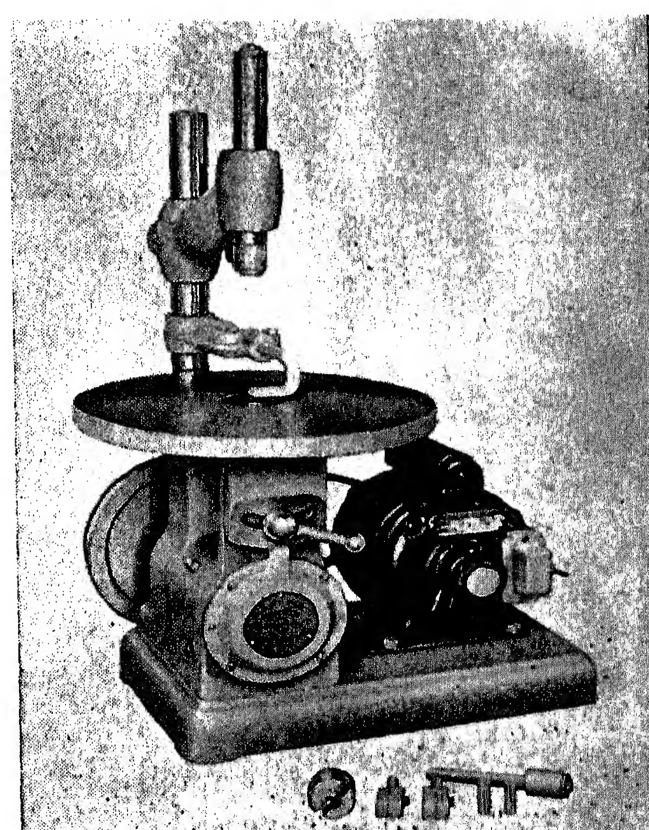


Fig. 15-7. A die filing machine with a 1½ in. stroke and 12 in. diameter table. (Courtesy Illinois Tool Works.)

pushed by hand into the saw. Gates and risers can be cut off of castings in this way. For heavy work, mechanical, pneumatic, or hydraulic feeds are available. Contour sawing machine tables can be tilted for cutting inclined surfaces.

An example of contour work done on a band saw machine is given in Fig. 15-6. Narrow band saws are used for curved surfaces. The progress of a cut is readily controllable and can be guided to follow a line. For an internal opening, a hole is first drilled in the piece. The saw band is broken, passed through the hole, and butt welded together on a jig. A small grinding wheel smooths the blade at the weld. A butt welding and grinding attachment is shown on the side of the machine column behind the table in Fig. 15-5. When the cut is finished, the blade is recut and removed from the opening in the workpiece. Often the slug sawed out can be used for a mating part, such as for the punch of a die.

An *open end band sawing machine* is made especially for internal work. One end of a saw band 140 feet long is slipped through a hole in the workpiece. The band is pulled through the work and is wound helically over a drum. At the end of the cut, the machine stops, and the saw band is rewound to starting position.

Contour band sawing machines are adaptable to other operations. A continuous band of file segments may be put in place of the band saw. Stones are used for hard materials. An abrasive belt may be used for polishing. Friction sawing is done on high speed machines with bands having dull or no teeth.

Die sawing and filing machine. Machines like the one in Fig. 15-7 are used for filing, sawing, and stoning. The tool is held at both ends and reciprocated through a hole in the center of the table. A 1½ in. stroke and 382 or 500 strokes per minute are available. Roller supports for a file and a guide for a saw are applied just above the table. The 12 in. diameter table can be tilted from 0 to 20 degrees on each side of center.

Saws and Files

Hack saw blades. The important features of a hack saw blade are (1) material, (2) tooth form, (3) tooth set, (4) tooth spacing, (5) blade thickness, (6) blade width, and (7) blade length.

Materials of which good quality hack saw blades are made are (1) tungsten high speed steel, (2) molybdenum high speed steel, (3) flexible tough alloy backing with high speed steel teeth, and (4) semihigh speed steel.

The most common form of hack saw tooth has a straight 56° back angle as shown in Fig. 15-8 A. Some saws with coarse pitches, like 2 or 2½ teeth per inch, have shapes like that of Fig. 15-8 B.

Saw teeth are offset to the sides to make the cut wider than the thickness of the back of the blade to prevent rubbing. The width of the slot is called the kerf. Three common types of saw settings are shown in Fig. 15-9. The *raker tooth* set has each straight tooth followed by two teeth set in opposite directions. The *wave tooth* set has one set of waves to the right, the next to the left, etc. In the *straight tooth* set all teeth are offset alternately to right or left.

Tooth spacing is expressed by the number of teeth in each inch of length of a saw blade. Six tooth spacings are common in the range from 2 to 14 teeth per inch. The length of cut determines the desirable tooth spacing as indicated in Fig. 15-10. In general, as coarse a tooth as feasible is recommended but at least two or three teeth should be in contact with the work. Some hacksaw blades are made with fine teeth on the starting end.

Power hack saw blades are made in five thicknesses from 0.032 to 0.100 in., in six widths from ⅝ to 2½ in., and in nine lengths from 12 to 32 in. Thick and wide blades are necessary for heavy cuts. A hack saw blade should be about twice as long as the maximum length of cut for workpieces over 6 in. in diameter.

Speeds from 50 strokes per minute for hard and tough materials to 150 strokes per minute for soft metals are recommended

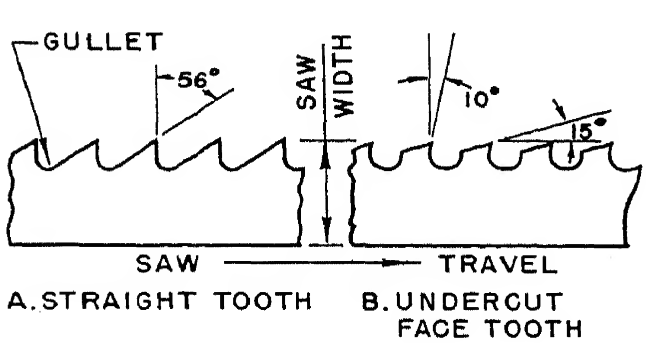


Fig. 15-8. Hack saw tooth angles.

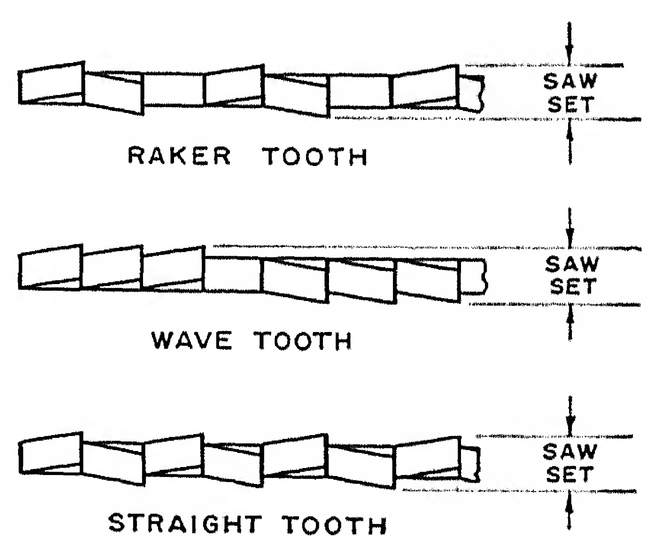


Fig. 15-9. Types of saw tooth sets.

for power hack sawing. The rate at which a hack saw cuts depends upon the feeding pressure exerted upon it. The pressure should be as large as possible without damaging the saw and generally runs from 10 to 50 lb for small pieces to 200 to 300 lb for large pieces.

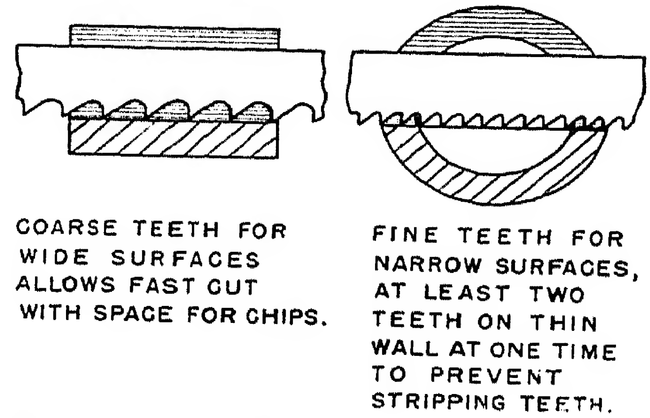


Fig. 15-10. The significance of saw tooth spacing.

Circular cold saws. The important features of a cold saw are (1) tooth material, (2) tooth form, (3) pitch, and (4) diameter.

Most cold saws have high speed steel teeth, but some are made with cemented carbide teeth.

Cold saws have three forms: solid blades, segmental blades, and inserted tooth blades. A solid blade has teeth cut directly in the disk and generally is not over 18 in. in diameter. Segmental blades are made up of a series of segments, each with several teeth, located by a tongue and groove mounting, and riveted on the saw disk, as indicated in Fig. 15-11. Inserted teeth are placed in slots around the saw blade disk and held by wedges or brazed. The teeth can be replaced individually but cannot be spaced as closely together as solid or segmental teeth.

A typical cold saw tooth is cam generated to the contour shown in Fig. 15-11. The curve ends at the top of the cutting edge with an angle of 7 to 11°. A flat 1/16 to ⅛ in. long with a rake angle of 12 to 25° is provided below the edge. Small rake and clearance

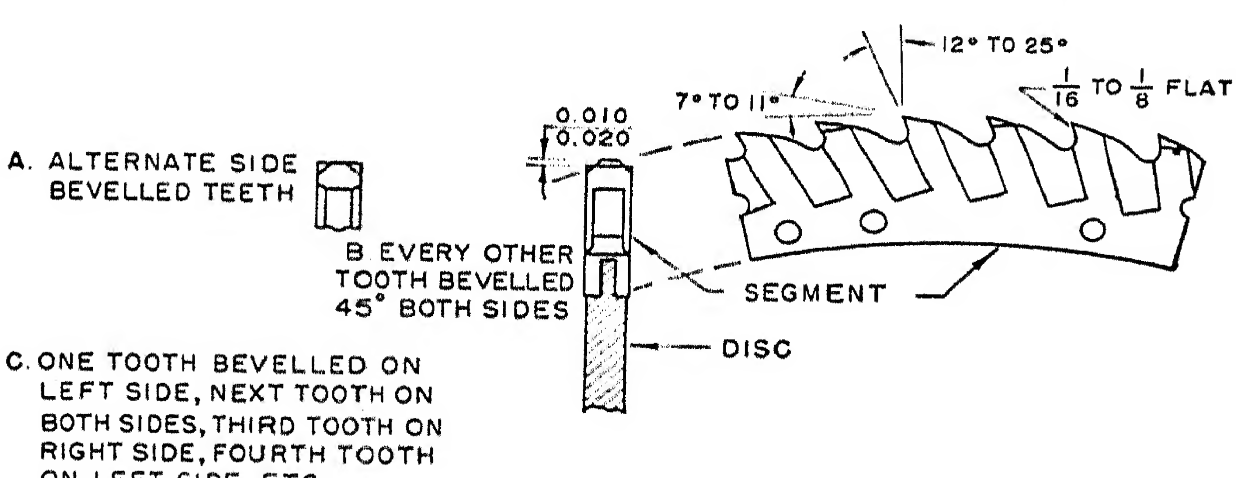


Fig. 15-11. Circular saw tooth profiles.

angles are desirable for hard materials, larger angles for soft materials. A well-rounded and smooth gullet helps the chips to curl and eases the cutting action. Circular saws often have alternately high and low teeth. Teeth may be beveled in various ways indicated in Fig. 15-11. These profiles help break up the chips, distribute the load, and allow some teeth to take finishing cuts.

The pitch of a circular saw is the distance from one tooth to the next. It may vary from 0.20 in. for small saws for tough materials to 2 in. on large diameter saws for soft materials. Enough space must be provided between teeth to furnish ample room for the chip removed during a cut. A wire brush or a spoked wheel may be provided to run along the saw to push chips from between the teeth.

The outside diameter of a circular saw must be large enough for one side of the saw to pass through the work. Cold saws range in diameter from about 8 to 36 in.

Band saws. The important features of a band saw are (1) material, (2) heat treatment, (3) tooth form. (4) tooth set, (5) tooth spacing, and (6) blade width.

Band saw blades are made from high speed steel for cutting the hardest materials and high carbon alloy steels for general work and soft materials. Some saws are hardened throughout, but others are tempered in back of the teeth for flexibility. Tempering is particularly desirable for blades for contour sawing.

The form of band saw teeth for general metal cutting is similar to that of other saws. Many other forms of blades are available for other materials and specific operations. Among these are scallop-edge band saw blades for soft fibrous materials like cloth, spiral saw blades that cut in all directions, knife edge blades for paper and fabrics, and diamond tooth blades for ceramic and vitreous materials.

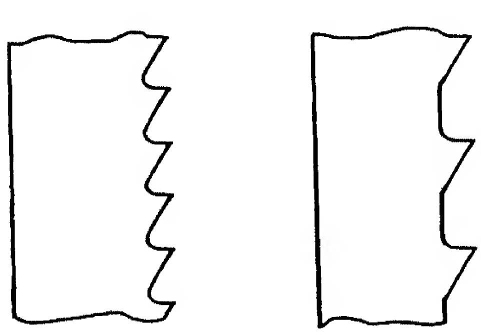


Fig. 15-12. Band saw teeth.

Raker tooth band saws are used for cutting iron and steel except in thin sections. The wave tooth gives the smallest possible tooth spacing and is desirable for thin sheets and sections. The straight tooth is applied to nonferrous metals and nonmetals.

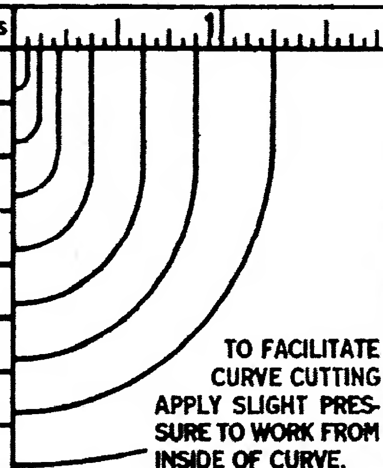
Tooth spacing of band saws ranges

SELECTION OF SAW BLADES

WIDTH OF BLADE

THE WIDTH OF THE BLADE IS DETERMINED BY THE SMALLEST RADIUS TO BE CUT. USE AS WIDE A BLADE AS POSSIBLE; A WIDER BLADE IS STRONGER AND CUTS MUCH FASTER.

WIDTH OF BLADE	SMALLEST RADIUS
1/16	1/16
3/32	1/8
1/8	7/32
3/16	3/8
1/4	5/8
5/16	7/8
3/8	1 1/4
1/2	3



STANDARD SPECIFICATIONS

FLEXIBLE BACK METAL CUTTING SAW BLADES ARE FURNISHED IN THE FOLLOWING PITCH:

WIDTH OF BLADE	TEETH PER INCH
1/16	24-32
3/32	24-32
1/8	18-24-32
3/16	12-14-18-24-32
1/4	10-12-14-18-24-32
5/16	10-12-14-18-24-32
3/8	8-10-12-14-18-24-32
1/2	8-10-12-14-18-24-32

USE RAKER TOOTH FOR ALL METALS

TEETH PER INCH OF BLADES

THE NUMBER OF TEETH IS DETERMINED BY THE THICKNESS OF STOCK. FOR FAST CUTTING USE AS COARSE A PITCH AS POSSIBLE.

THICKNESS OF STOCK	TEETH PER INCH
0 - 3/32	32 - 24
3/32 - 3/16	24 - 18
3/16 - 3/8	18 - 14
3/8 - 3/4	18 - 14 - 12
3/4 - 1 1/4	14 - 12 - 10
1 1/4 - 2	12 - 10 - 8
2 - UP	10 - 8

SPEED OF BLADES

SPEEDS	F.P.M.	SPEEDS	F.P.M.
①	50	⑥	390
②	75	⑦	585
③	115	⑧	900
④	175	⑨	1365
⑤	260	⑩	2030

MATERIAL	SPEED	MATERIAL	SPEED	MATERIAL	SPEED	MATERIAL	SPEED
		BAKELITE	4-8	MALLEABLE IRON	2-3	STRUCTURAL STEEL	2-3
ALUMINUM	4-10	CAST IRON	1	METAL WOOD	4-10	STEEL SHEETS	3-4
AMPCO METAL #16-18	2-3	COPPER	3-4	MATRIX METAL	5-8	TUBING-ALUM.	5-10
AMPCO METAL #18-20	1-2	C. R. STEEL	2-3	MICA	4-8	TUBING-BRASS	4
ASBESTOS SHEETS	3-4	DRILL ROD	1-2	MONEL METAL	1-2	TUBING-STEEL	2-3
BABBITT	4-8	FIBRE	3-4	NICKEL SILVER	1-2	TOOL STEEL-AIR HD.	1
BRASS CSTGS-HARD	2-3	GOLD	3-4	NICKEL STEEL	1-2	TOOL STEEL-OIL HD.	1-2
BRASS CSTGS-SOFT	4-6	H. CHR. HIGH CARBON ST.	1	RUBBER-HARD	4-6	TOOL STEEL-WATER HD.	2-3
BRASS SHEETS	4-8	HIGH SPEED STEEL	1	SILVER	2-3	WROUGHT IRON	3-4
BRONZE CASTINGS	2-3	IRON SHEETS	3-4	SLATE	1-2	WOOD	5-10
BRONZE MANGANESE	2-3	MACHINE STEEL	2-3	STAINLESS STEEL	1	ZINC	4-8

GROB BROTHERS GRAFTON, WIS., U.S.A.

Fig. 15-13. Typical recommendatons for selection and operation of band saws.

from 2 to 32 teeth per inch. The medium and finer pitches have *standard pitch construction* indicated in Fig. 15-12. Coarse pitches, from 2 to about 6 teeth per inch, have the *buttress* or *skip tooth construction* of Fig. 15-12 for soft metals and nonferrous materials. In general, a relatively coarse pitch is selected for thick material and a fine pitch for thin material. Typical recommendations of one manufacturer are given in Fig. 15-13.

Band saws are made in widths from 1/16 to 1 in. The radius that can be cut in contour sawing depends upon the width of the saw band, as depicted in Fig. 15-13.

Operation of saws. Substantially the same considerations govern the speeds and feeds of sawing as apply to other processes. Cold sawing is comparable to milling, and the proper speeds and feeds for cold saws are like those for similar milling cutters. Typical speeds for band sawing are specified in Fig. 15-13. The speed of a hack saw is usually given in strokes per minute but is basically determined by the surface speed of the cutting stroke.

The feed and length of cut in sawing determine the rate of stock removal and the consequent load on the saw, machine, and workpiece. One manufacturer recommends feeds from 6 in. per minute for ¼ in. thick stock to 3/16 in. per minute for 6 in. thick stock for straight cutting of machine steel with a ¼ in. wide band saw. Harder materials call for lower feeds, softer materials permit higher feeds. A wider blade cuts faster, but a narrower blade cuts slower. Contour cutting must be done at a slow feed to produce a uniform and true curved surface.

Files. Files are identified by (1) method of application, (2) class, (3) cut, (4) pitch, and (5) size.

The common file with a tang is usually operated by hand, with a wooden handle over the tang. Such files may be used on a die filing machine like the one in Fig. 15-7. The file is reciprocated across the work, with pressure applied on the forward stroke and released on the return stroke.

Continuous filing is done with a band file, which is made up of a series of short file segments. Each segment is fastened near its leading end by a clip to a flexible steel band so that it can pass freely over the wheels on which the band runs. Each file segment is interlocked and fits snugly with the segment behind it so that the cutting action is not interrupted. The ends of the band are

connected by a latch so that they can be separated, one end can be passed through an opening in a workpiece, and the ends can then be rejoined.

Band filing is not meant to replace hand filing entirely but is useful for removing the bulk of stock, leaving little for the arduous hand process.

The cross-sectional shape of a file determines its *class*. About 20 shapes are recognized as standard. The principal shapes are rectangular (flat), square, part round (half round), round, oval, and triangular (three cornered). Band files commonly have flat, half round, oval, and three cornered shapes.

The *cut* of a file designates the way the teeth are cut. A *single cut file* has single rows of parallel teeth across its face at an angle with its axis. A *double cut* file has two rows of teeth crossing each other, one row finer than the other. Each tooth of a *rasp cut* file is formed by itself by a single punch mark. A *vixen cut* is a milled cut with large knifelike teeth, often curved across the face of the file.

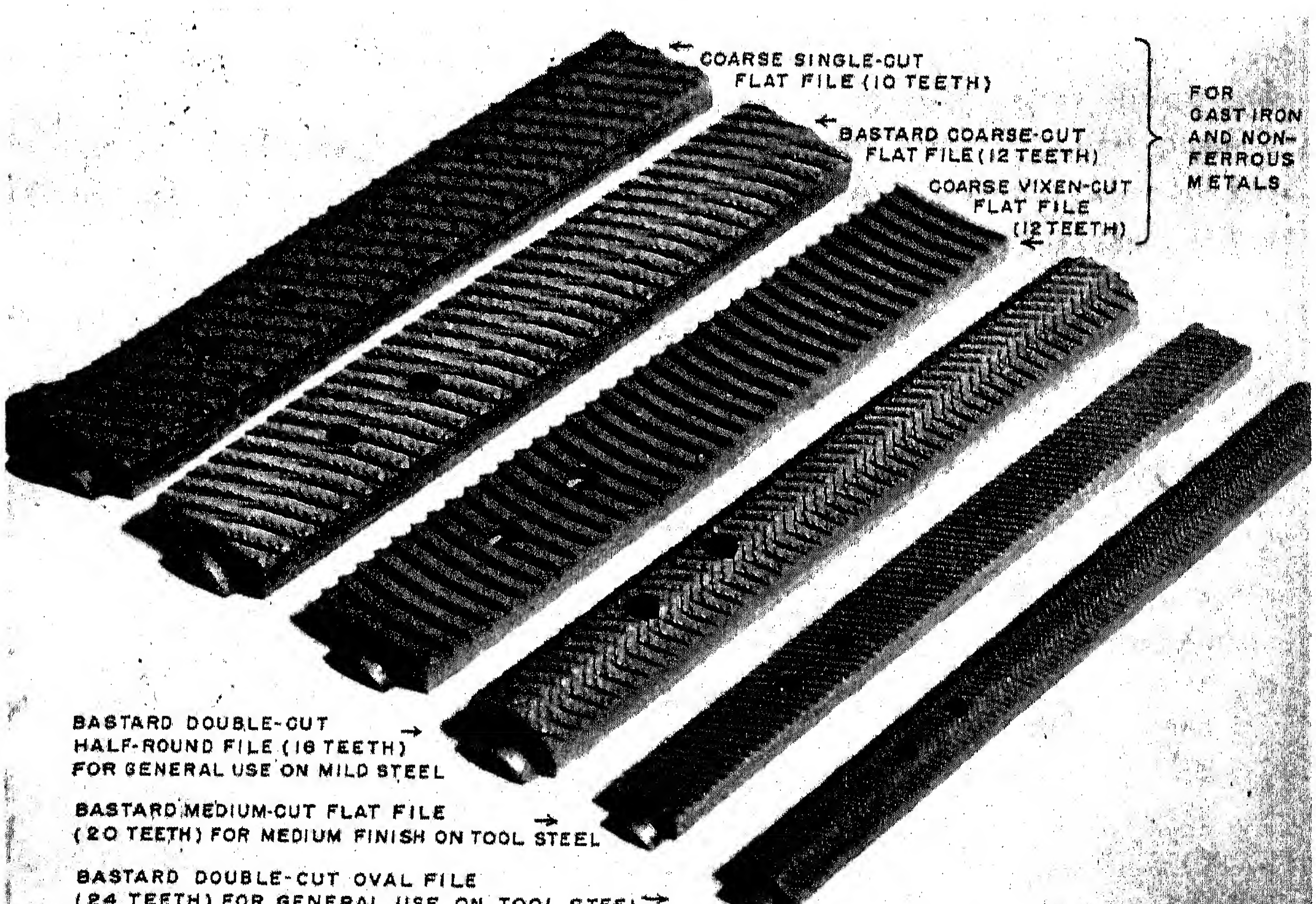


Fig. 15-14. Typical band file segments. (Courtesy The DoAll Co.)

Files are graded according to the pitch or spacing of the teeth. Descriptive names that refer to the pitch, in order from wide to close spacing, are rough, coarse, bastard, second cut, smooth, and dead smooth. The spacing may be designated by the number of teeth per inch, but no relationship between the number of teeth and the descriptive names is universally accepted. Some files are designated by a series of numbers from the coarsest, No. 00, 0, and 1 through 8, the finest.

Several typical band file segments of various shapes, cuts, and pitches are shown in Fig. 15-14.

Hand files are available in many widths up to over one inch. Band files are commonly ¼, ⅜, and ½ in. wide. The specified length of a hand file does not include the tang. Lengths range from 2 to 14 in., but the most common are 10 to 14 in.

Coarse cut files remove metal rapidly; fine cut files give good finishes. Medium cut files offer a compromise between rapid metal removal and finish. Files most commonly used for ferrous metals are flat or half round; double or single cut; in bastard, second cut, and smooth grades. Files should have deeply cut teeth for chip clearance for nonferrous metals and nonmetals.

Band filing speeds range from 50 sfpm for harder steel alloys to 250 sfpm for soft materials. A heavy pressure is applied for rapid metal removal, a light pressure for finishing.

Questions

1. What advantages does sawing offer over other metal cutting processes?
2. What advantages and limitations does filing have?
3. How does a power hack saw machine work?
4. What are the advantages and disadvantages of hack sawing?
5. What advantages does circular sawing offer for cutting off pieces in production?
6. What two kinds of band saw machines are available? For what are they used?
7. How may internal openings be band sawed?
8. Discuss the important features of hack saw blades.
9. Describe the important features of a cold saw.
10. Discuss the important features of a band saw.
11. What is meant by the class, cut, and pitch of a file?

References

.e Circular Saw. Fitchburg, Mass.: Simmonds Saw and Steel Co., 1929.

'cular Sawing Handbook. Cleveland, Ohio: The Motch and Merryweather Co., 1950.

) All Contour Saws. Des Plaines, Ill.: Do All Technical Institute, 1947.

oor, W. T., "Factors Important to Low Cost Circular Sawing," *Machinery*, April, 1950.

ıck Saws and Their Uses. Athol, Mass.: L. S. Starrett Co., 1931.

anufacturers' Catalogs:
Armstrong-Blum Mfg. Co., Chicago, Ill.
The Do All Co., Des Plaines, Ill.
Grob Bros., Grafton, Wis.
Illinois Tool Works Co., Chicago, Ill.
Johnson Mfg. Co., Albion, Mich.
Motch and Merryweather Machinery Co., Cleveland, Ohio.
Wells Mfg. Co., Three Rivers, Mich.

Chapter 16

BROACHING MACHINES AND BROACHES

BROACHING IS A MACHINING OPERATION wherein a tool with a series of teeth called a *broach* is pushed or pulled on a broaching machine along a surface of a workpiece. Each tooth takes a thin slice from the surface. Broaching of inside surfaces is called *internal* or *hole broaching;* of outside surfaces, *surface broaching.*

Broaching Machines

Types and sizes of broaching machines. Broaching on a small scale is often done on manual arbor presses, but power-driven broaching machines are used for production. Mechanical drives, utilizing a rack and pinion or screw and nut, were common on old machines, but most modern machines have hydraulic drives.

Several types of broaching machines can accommodate most work, and they have become recognized as standard. They are:

1. Broaching presses
2. Horizontal pull broaching machines
3. Vertical pull down or pull up broaching machines
4. Vertical single or double ram surface broaching machines
5. Horizontal surface broaching machines
6. Horizontal and rotary continuous broaching machines

The size of a broaching machine is designated by the length of stroke in inches and the force in tons that can be applied to the broach. Capacities range from a fraction of a ton up to 50 tons in general use. The commonest sizes are from 10 to 20 tons. Strokes are available up to 90 in.

Broaching presses. Push broaching and some surface broaching are done on various kinds of presses. A few pieces at a time may be broached on a hand-operated arbor press. Hydraulic arbor presses with extra equipment and hydraulic presses specifically arranged for broaching, called *push broaching machines,* are available for production. Presses range in capacity from 500 lb to 35 tons.

A 6 ton hydraulic arbor press is shown in Fig. 16-1 with a guided ram, controlled ram speed, and coolant system to adapt it to broaching. Work is placed on the table, sometimes in a simple fixture, and the vertical ram pushes a short broach through the work.

Broaching presses are used mostly for internal broaching. Hole sizing and keyway cutting are typical operations. Push broaches must be relatively short, and each one cannot remove much stock. Some surface broaching is done, but it requires fairly elaborate tooling. The advantage of push broaching is that operations can be set up and changed over easily.

Arbor presses are popular for push broaching because they are simple and inexpensive and can be readily used for many other operations such as assembling, bending, drawing, and staking. Thus the capital charge for any one operation can be quite small.

Horizontal pull broaching machine. An internal broach is being pulled through a workpiece on a horizontal pull broaching machine in Fig. 16-2. The workpiece is located in a cup centered in the machine face plate at the back of the picture.

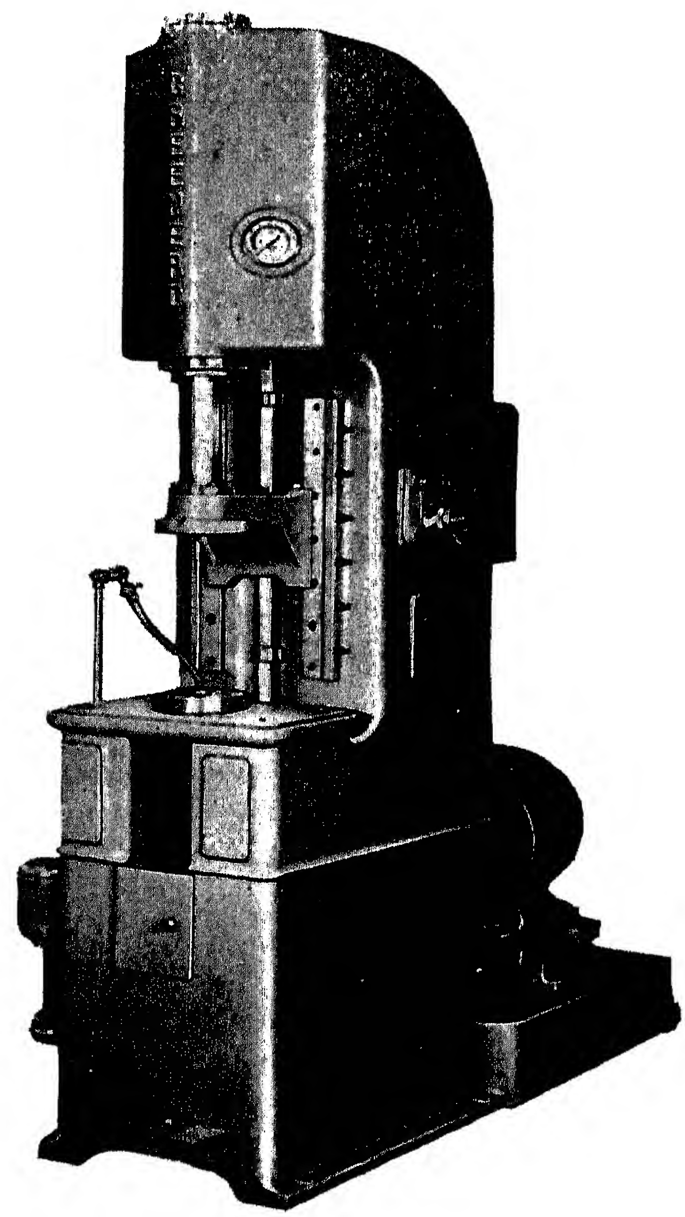


Fig. 16-1. A 6 ton hydraulic arbor press with guided ram, controlled ram speed, and coolant system for broaching. (Courtesy Greenerd Arbor Press Co.)

The broaching force presses the piece against the face plate. Various simple means are used on these machines to hold workpieces. As another example, a piece in which an internal keyway is broached may be slipped over a plug having a lengthwise slot to guide the narrow broach. More elaborate fixtures may be needed to support heavy or intricate pieces.

The broach is pulled by a horizontal ram actuated by a hydraulic piston and cylinder, behind the face plate and not shown in Fig.

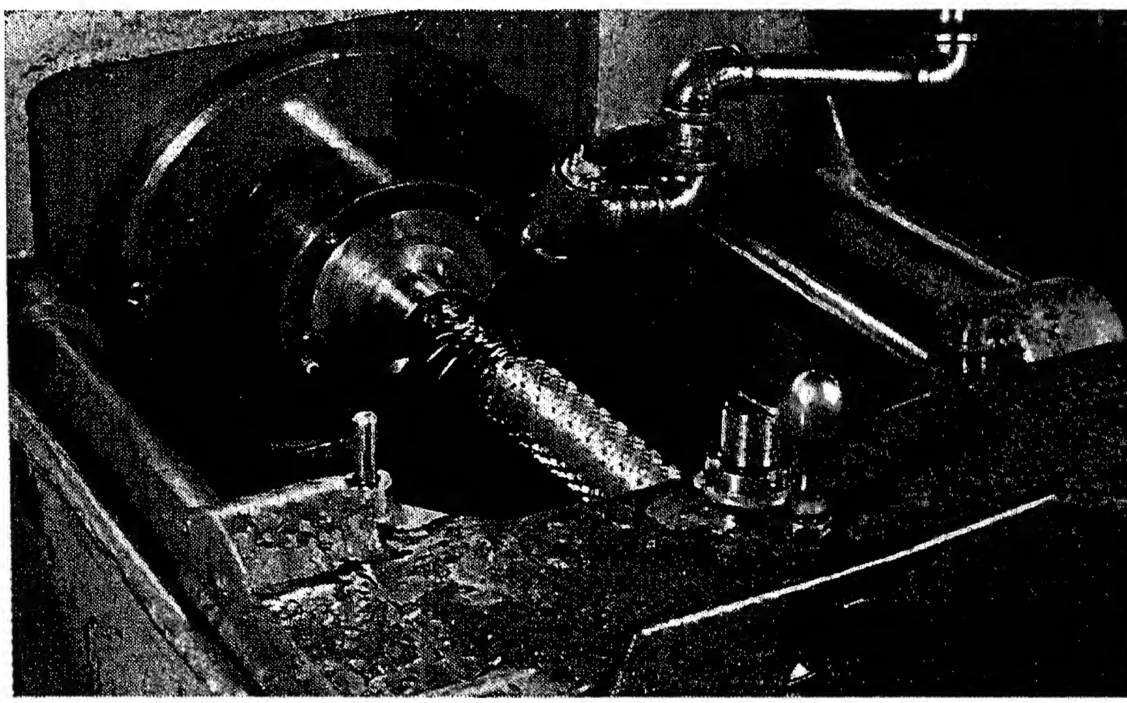


Fig. 16-2. An internal helical spline being broached on a horizontal pull type broaching machine. (Courtesy Colonial Broach Co.)

16-2. The operator passes an internal broach through the initial opening in the workpiece and attaches it to the drawhead. The outer end of a heavy or long broach is supported by an outer support like the one in the foreground of Fig. 16-2. The outer support moves toward the work with the broach and also serves to return the broach to its starting position after each piece is finished and removed. Many horizontal broaching machines are arranged to handle broaches semiautomatically or fully automatically.

Machines of this type are commonly used to broach round and other shapes of holes, keyways, and splines. The helical spline broached in Fig. 16-2 requires a special broach puller to rotate the broach as it passes through the work.

Horizontal pull broaching machines also handle many surface broaching jobs efficiently. An example is given in Fig. 16-3 by the machine set up to broach three flats on the periphery of a disk. A fixture is provided to hold the workpiece and guide the broach bars. A surface broach is not normally disconnected from the draw-head when each piece is loaded.

A horizontal broaching machine is convenient because access to any part of the machine is easy. Heavy broaches do not have to be raised and lowered. A long stroke is feasible without necessitating

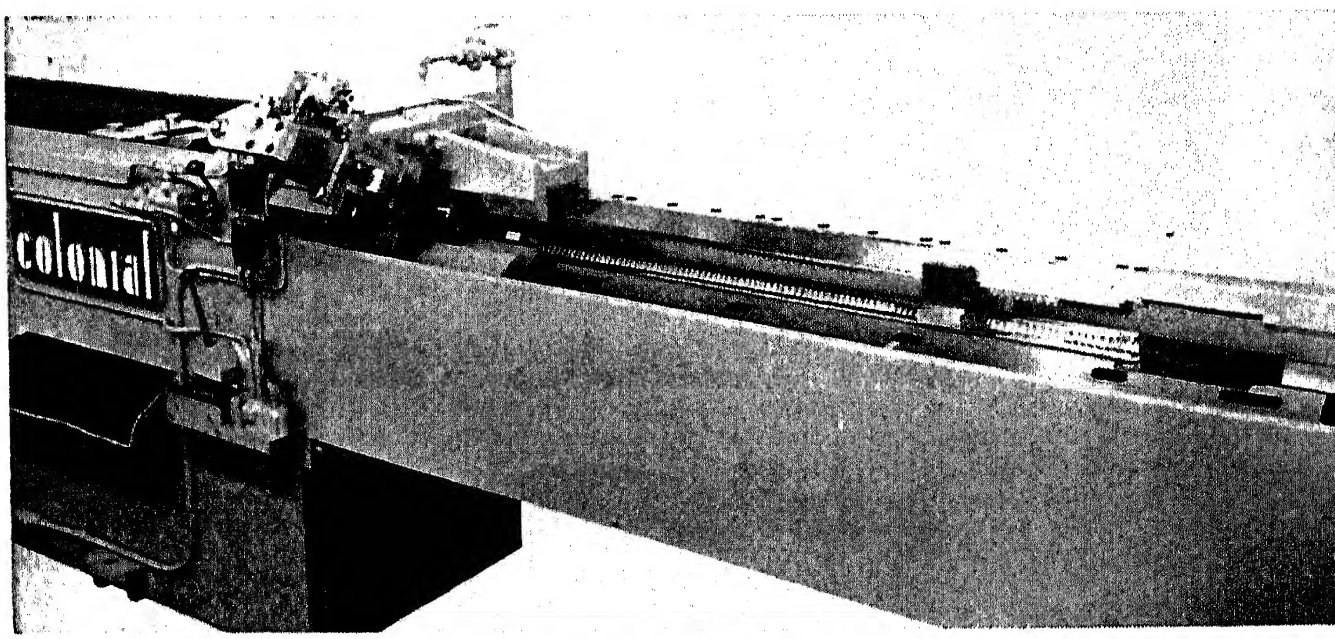


Fig. 16-3. A horizontal pull type broaching machine set up for broaching three flats on the outside of a disc. (Courtesy Colonial Broach Co.)

a tall machine with the work station high in the air, as with large vertical broaching machines, but more floor space usually is needed for a horizontal machine.

Vertical pull broaching machines. The *vertical pull down broaching machine* of Fig. 16-4 is arranged for broaching holes in two crank arms. The pieces are placed in fixtures on the table of the machine. The operator presses the two buttons, one on each side of the table, to signify his hands are free and to start the machine. The broaches are lowered into the holes in the parts by the elevator from which they are suspended. As soon as their pilots pass through the work, the broaches are automatically grasped by puller heads on a ram below the table and released at the top. The lower ram pulls the broaches through the parts. At the end of the stroke, the

parts are removed, and the broaches are brought back up to the elevator which raises them above the table so new parts can be loaded for the next cycle.

Some pull broaching machines can be arranged with a head at the top of the ram for push broaching. Broaches can also be mounted on the face of the ram for surface broaching.

Vertical pull down broaching machines are made with one, two, or more broaching stations and are capable of high rates of production. Large and irregular pieces can be set in place and can be loaded somewhat more easily than on pull up machines. Fixtures can be used to hold the locations of broached holes in relationship with external surfaces and contours of parts. Application of cutting fluid is simple and effective because the flow is in the direction of broach travel.

The *vertical pull up broaching machine* of Fig. 16-5 is arranged for two station internal broaching utilizing a semiautomatic cycle.

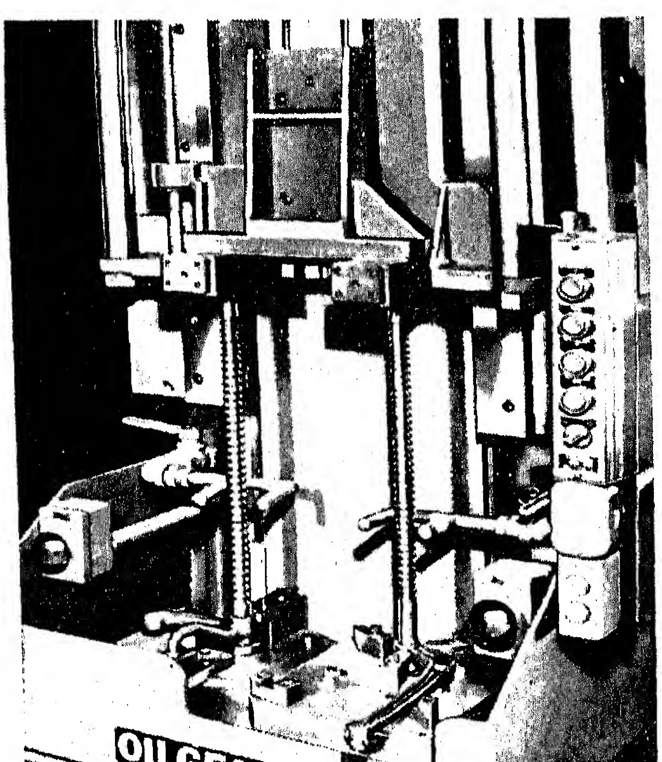


Fig. 16-4. A view of a set up on a vertical pull down broaching machine. (Courtesy The Oilgear Co.)

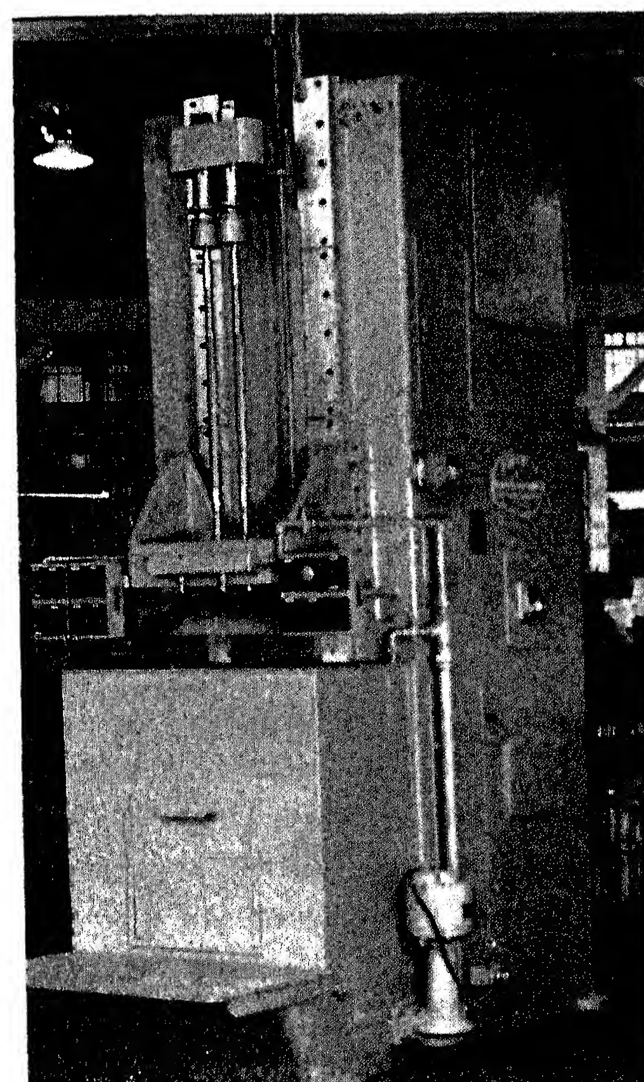
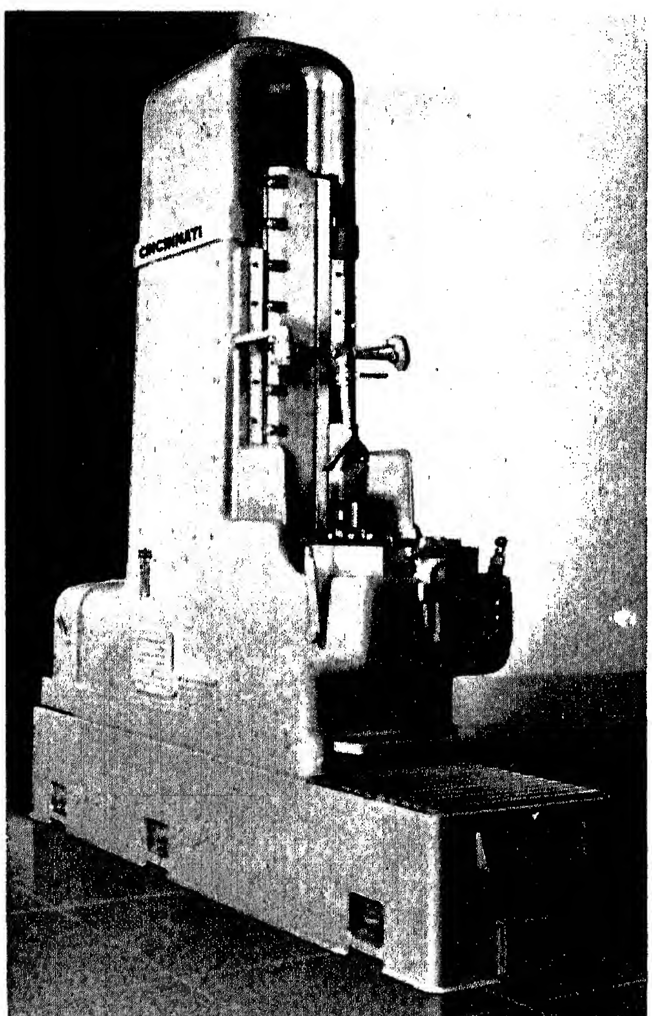


Fig. 16-5. A 15 ton vertical pull up broaching machine with a 48 in. stroke. (Courtesy American Broach and Machine Co.)

The broaches are shown engaged with the pull heads on the ram but are lowered below the table and released from the heads to start a cycle. Parts are placed over the pilots, and an elevator below the table raises the broaches to where they can be grasped by the pull heads. The pieces are pressed against the underside of the table, and the broaches are pulled through them. At the end of the stroke, the parts fall down a chute at the front of the machine.

Vertical pull up broaching machines are high production machines. They commonly are made to pull four broaches at a time, sometimes as many as eight. The operator's motions are saved because the broaches are handled entirely mechanically and the workpieces are discharged automatically.



Fif. 16-6. A 3 ton vertical single ram surface broaching machine with 48 in. stroke. (Courtesy The Cincinnati Milling Machine Co.)

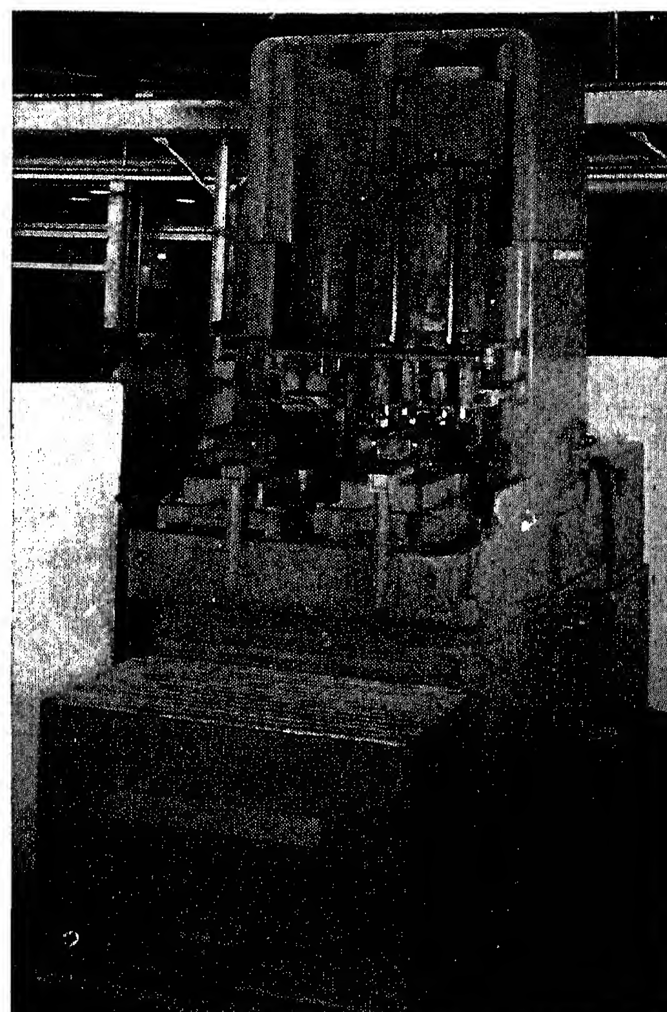


Fig. 16-7. A 10 ton vertical duplex surface broaching machine with a 66 in. stroke. (Courtesy The Cincinnati Milling Machine Co.)

Vertical surface broaching machines. The *vertical single ram surface broaching machine* of Fig. 16-6 has a ram on the front of the column for mounting a broach. The ram carries the broach downward for the cut and returns to top position at twice the cutting speed. The table or platen is interlocked with the ram movement, goes into cutting position just before the ram descends, and is withdrawn on the upward stroke. Work fixtures are mounted on the table and are unloaded and loaded safely while the table is retracted and the ram is ascending. Some machines have a fixed table. On those machines the ram is stopped at the bottom and top of its stroke for unloading and loading the work.

Vertical surface broaching machines are designed so that work can be done quickly and conveniently on them. They occupy a minimum of floor space.

The *vertical duplex or double ram broaching machine* of Fig. 16-7 has two rams and tables that alternate. One ram descends to cut while the other rises to the starting position. The fixture in front of the descending ram is moved into cutting position while the other fixture is retracted so that it can be unloaded and loaded. These machines quite often operate continuously.

A double ram surface broaching machine is equivalent to two single ram machines. Its rate of production is high because it can be made to cut practically all the time. Often two operations on one part are done on one machine.

Horizontal surface broaching machines. Horizontal surface

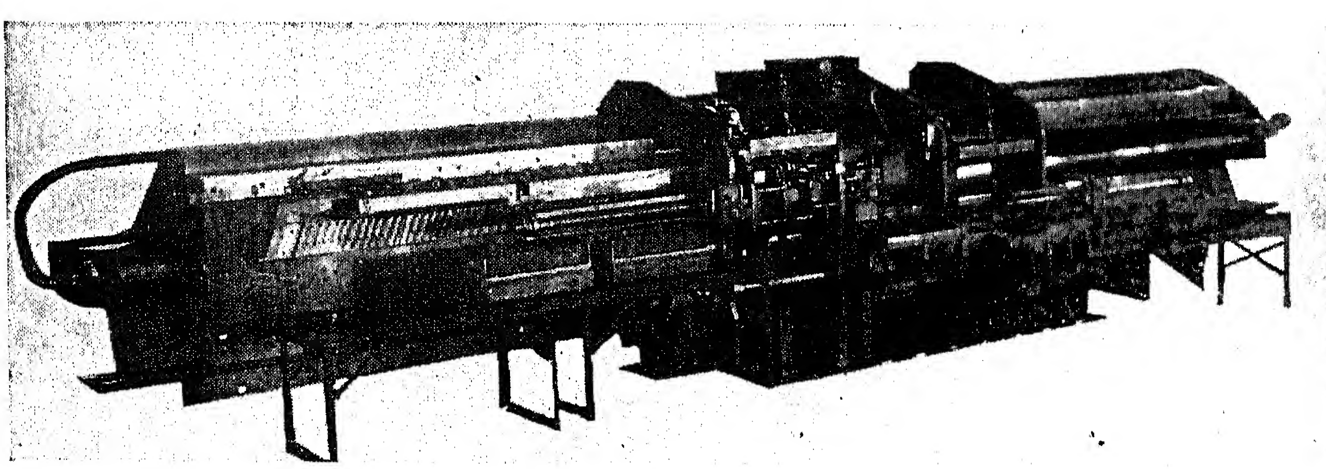


Fig. 16-8. A 25 ton horizontal double acting horizontal surface broaching machine for finishing the bottom, top, valve chamber cover face, and distributor pad on a six cylinder engine block at the rate of 60 blocks per hour. (Courtesy The Cincinnati Milling Machine Co.)

broaching machines have been developed for parts too bulky or heavy to handle conveniently on vertical surface broaching machines. The typical horizontal machine illustrated in Fig. 16-8 is double acting. Two sets of broaches are mounted on its ram and cut in both directions. Some machines are only single acting.

The height of a horizontal surface broaching machine is independent of the length of stroke of the ram, and these machines commonly have long strokes, in excess of 66 in. These machines can be integrated readily with waist-high roller conveyor systems. Workpieces are brought on a conveyor to the first of the two fixtures on the machine illustrated. The fixture swings into loading position and then carries the workpiece to cutting position. One fixture holds a workpiece for the forward stroke of the ram, and the other for the return stroke, and they are loaded alternately. A transfer mechanism between the two fixtures turns the blocks over for loading in the second station.

Continuous broaching machines. The horizontal continuous broaching machine of Fig. 16-9 is different from the machines previously described because the broach cutters mounted on it remain stationary while the workpieces are traversed. At the right-hand end of the machine, workpieces are loaded in a series of fixtures carried by an endless chain. The pieces are carried past the broaches, are released at the other end of the machine, and are discharged down a chute. Parts may be broached as rapidly as they can be loaded.

One type of rotary continuous broaching machine is somewhat

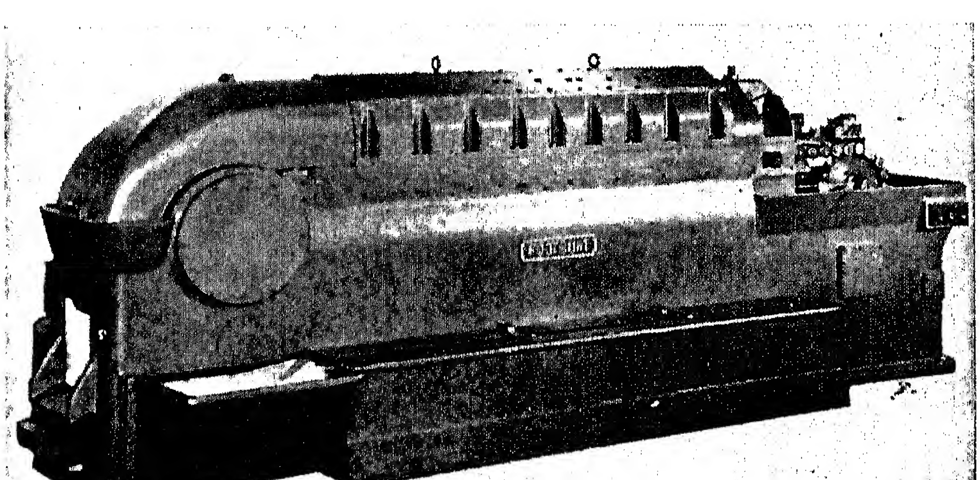


Fig. 16-9. A horizontal continuous broaching machine. (Courtesy The Foote-Burt Co.)

like a rotary miller. It has a slowly revolving round table carrying a group of fixtures in a circle. Workpieces are loaded in the fixtures and carried past stationary broaches. On some machines the fixtures are unclamped and the parts ejected automatically.

Broaches

Types of broaches. A broach is a cutting tool with multiple teeth, each designed to remove a certain amount of stock during the cutting stroke. Broaches may be classified according to:

1. ways they are operated: push, pull, or stationary
2. kinds of operations they perform: internal and external

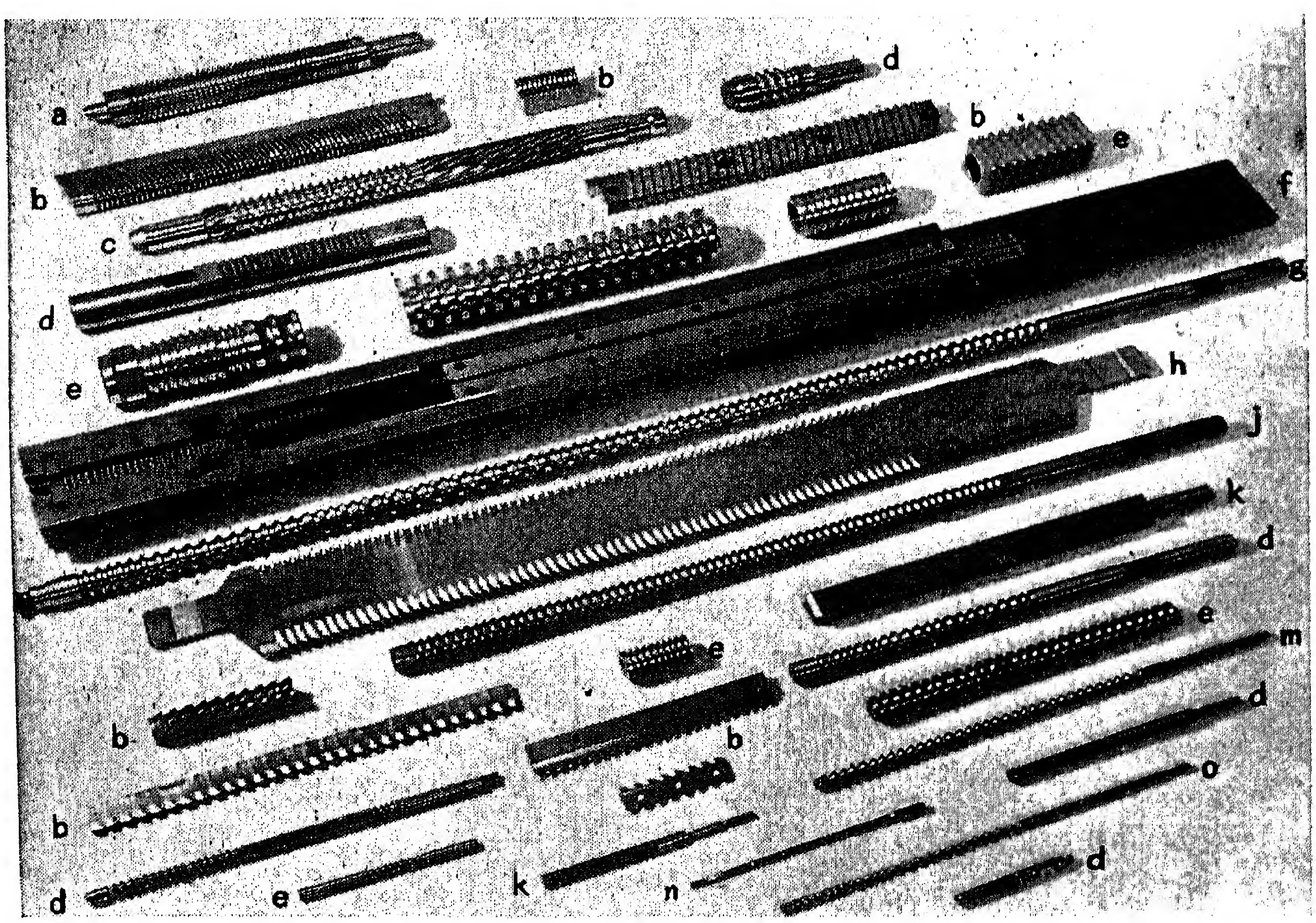


Fig. 16-10. Typical broaches. (Courtesy Continental Tool Works, Ex-Cell-O Corp.) a. Combination hole sizing and involute spline push broach. b. Broach sections. c. Spiral spline hard gear pull broach. d. Push broaches. e. Broach shells. f. Broach inserts assembled in holder. g. Rotary cut pull broach for round hole. h. Slotting broach. j. Hole sizing and spline cutting pull broach. k. Flat broaches. m. Spline pull broach. n. Keyway broach. o. Combination hole cutting and burnishing pull broach.

3. their construction: solid, built-up, replaceable section, inserted tooth, progressive, overlapping tooth, etc.
4. their function: single purpose, combination, spline, keyway, burnishing, serration, rifling, etc.

The way a broach is used determines to a large extent its general proportions. Typical internal push and pull and external surface broaches are shown in Fig. 16-10. Push broaches are shorter than pull broaches to resist bending.

Most internal or hole broaches are of *solid construction,* but a few are built up of *shells* mounted on a bar or on the rear of a solid broach. Several kinds of shells are shown in Fig. 16-10. They are desirable where wear is rapid or extreme accuracy is necessary and the broach must be replaced often. A shell is cheaper than a whole new broach, but the first cost of a complete shell broach is higher than that of a comparable solid broach.

Surface broaches are commonly built up from sections, depicted in Fig. 16-10. In some cases the individual broach teeth can be removed and replaced. Broach inserts are mounted on a holder, to which they may be bolted or fastened by wedges, clamps, set screws, or locks. Inserts make a broach easier and cheaper to fabricate, alter, and sharpen. Descriptive names given to broaches of this kind are *built-up, replaceable section, sectional,* and *inserted-tooth broaches.*

Wide teeth on a surface broach may require too much driving force. This difficulty may be overcome by a *progressive broach* as indicated in Fig. 16-11. The first few teeth cut in the center. Succeeding teeth are offset to each side and progressively complete a wide surface as they pass over it. This broach must be made longer than one with wide teeth.

Forms of construction that are progressive in nature are found in round hole broaches. A round pull broach may have every other or every two alternate cutting teeth nicked to break up chips. A variation of that form is known as a *double cut broach.* Its teeth are arranged in sets of two of the same size. The first tooth in each set has wide chip breakers, and the second is fully round. Still another variation is the *rotary cut broach* of Fig. 16-10g. Each cutting tooth has a cutting edge for only part of the hole. The cutting edges of succeeding teeth are offset around the circum-

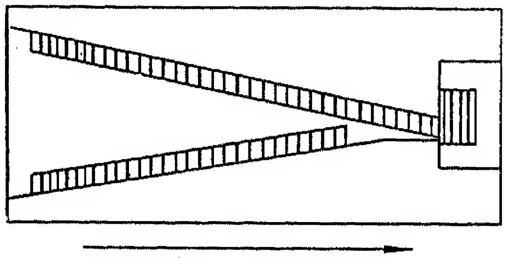


Fig. 16-11. The arrangement of a progressive surface broach.

ference to remove material missed by preceding teeth. Each tooth can take a relatively deep bite because it cuts only a small part of a circle, and the teeth can get underneath scale and inclusions. The tendency for a broach to drift to one side or another is reduced if its cut is interrupted.

A broach to cut just a round hole is a *single purpose broach,* as is one to cut just an involute spline. In contrast, a broach to cut both a spline and an inside diameter of a hole is a *combination broach.* Broaches may have various combinations. Some are shown in Fig. 16-10.

A *burnishing broach* makes a glazed surface in a steel, cast iron or nonferrous hole. Burnishing teeth are rounded and do not cut but compress and rub the surface metal. Broaches solely for burnishing often are short and of the push type. In some cases burnishing buttons are added to the end of a hole cutting broach.

Typical common applications of broaches are for cutting serrations, straight or helical splines, gun rifling, and keyways. A broach for such a specific purpose may be named for the operation it performs.

Details of broaches. Fig. 16-12 shows the details of a hole broach. The *shank* is the end in front of the teeth where the tool is gripped by the puller. The *front pilot* centers the broach in the hole before the teeth begin to cut. The first group of teeth remove most

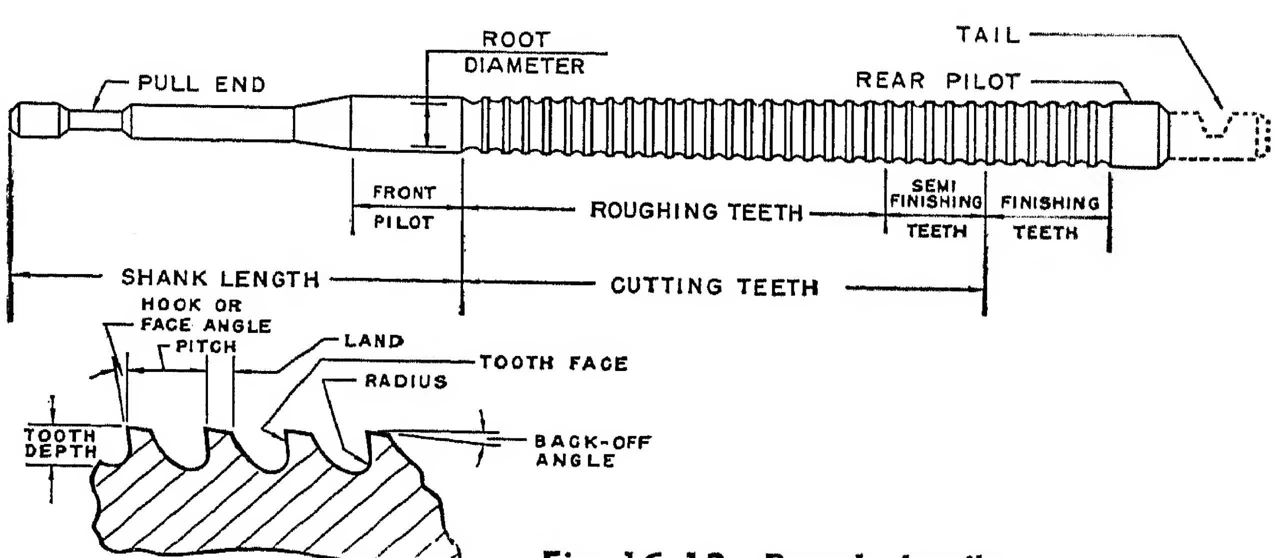


Fig. 16-12. Broach details.

of the stock and are called *roughing teeth.* The *finishing teeth* of a new broach are all the same size and must have exactly the shape required of the finished hole. As the first finishing teeth become worn, those behind take up the function of sizing. The *rear pilot* supports the broach after the last tooth leaves the hole. For close work it may be 0.001 in. to as much as 0.003 in. undersize. A broach handled automatically has a *tail* as indicated in Fig. 16-12.

The enlarged section of broach teeth in Fig. 16-12 reveals features like those of other cutting tools. The *back-off angle* on the outside of the teeth provides relief. The *hook or face angle* gives rake to the teeth, and its amount depends upon the material to be cut and is comparable to the rake angles of other kinds of cutting tools.

Each cutting tooth of a broach is larger than the one before it and smaller than the one after it. This step or difference in size is usually uniform in a series of teeth and determines the depth of cut taken by each tooth. The depth of cut determines the load on a tooth. Too heavy a cut means an overload. In general 0.003 to 0.006 in. per tooth is allowed for roughing free cutting steel. A cut should not be so light that a tooth rubs and does not cut, because rubbing ruins the cutting edge. Any step should be at least a few ten thousandths of an inch.

The spacing from tooth to tooth of a broach is the *pitch.* It must be long enough to provide sufficient space between the teeth for chips to collect during a cut. At the same time, at least two or three teeth must be in contact with the work at one time, so the pitch cannot be too large, especially for short holes or surfaces. A formula frequently used is that pitch $= 0.35\sqrt{\text{length of cut}}$.

The force required to push or pull a broach depends upon the step per tooth and the number of teeth engaged at one time under a given set of conditions. If the force required is greater than that available from the machine, it is considered better to reduce the step per tooth than to reduce the number of teeth in contact with the work by increasing the pitch. The smallest cross-sectional area of an internal broach must be enough to sustain the force imposed on the broach.

The number of cutting teeth in a broach is equal to the depth of stock to be removed divided by the step per tooth. The length of a broach is largely determined by the number of cutting teeth times the pitch per tooth. If the required broach length is greater than

the length of machine stroke, more than one broach must be used. Also, if the number of teeth required for a push broach dictates too long a broach, several push broaches are necessary.

Most broaches are made from high speed steel, ground after hardening. Carbon tool steel and carbon vanadium tool steel broaches are satisfactory for small quantities of parts. Some broaches have cemented carbide finishing teeth or rings to retain sizing ability over a long period of time. For hard materials, some roughing teeth may be made of cemented carbide.

The teeth of a *shear cut* surface broach lie at an angle, usually 5° to 20°, with the normal to the direction of cut. This is analogous to a helical milling cutter. Vibration is reduced, and a smooth cutting action and good surface finish are provided because the teeth enter and leave the cut gradually.

Broach sharpening. The cutting edges of the teeth of an internal broach are restored, after becoming dull, by grinding the faces of the teeth. Care must be taken that the smooth radius in the tooth space is retained. A thin dished wheel is used to get to the faces of the teeth without damaging other parts of the broach. For sharpening, the broach is revolved between centers. Sharpening may be done in a lathe with a toolpost grinder as a makeshift, on a cutter grinder, or on a universal grinder, but best results are obtained on cutter grinders made especially for the purpose and called *broach sharpening grinders.*

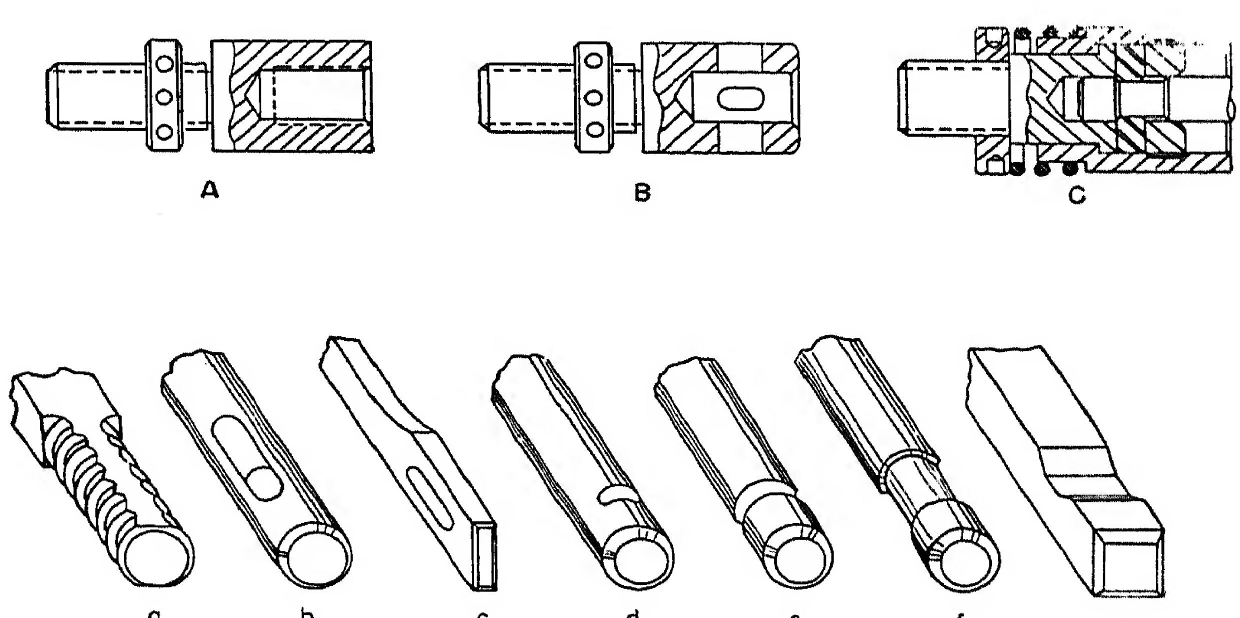


Fig. 16-13. Broach pullers and shanks.

Surface broaches having curved shapes may be sharpened like internal broaches. Other surface broaches are usually reground on the tops of the teeth with a taper over the length of the broach to provide the steps between the teeth. When an insert is taken off and ground on top, it is put back on shims in the broach holder to raise its cutting edges to their original position.

Broaches should be carefully handled and stored to avoid damage to them.

Broach Pullers and Fixtures

Broach pullers. Many surface broaches are mounted on holders bolted to the face of a ram, but some surface broaches and all pull broaches must be connected to the end of a ram by a *puller* or *puller head*. Several common pullers and broach shanks are shown in Fig. 16-13.

A *threaded puller*, like Fig. 16-13 A, is simple and inexpensive and may be used for keyway and other broaches that do not have to be disconnected from the ram for each pass. The threaded shank fits the machine ram. A typical broach shank to fit a threaded puller is shown in Fig. 16-13 a.

The *key-type puller* of Fig. 16-13 B is a simple and inexpensive device for pulling broaches that must be detached to be passed through each workpiece. Broach shanks in Figs. 16-13 b and c are typical of those used with key-type pullers. A key is placed through the puller and broach shank slots to attach puller and broach together. A broach too small to have a slot in its shank may have a groove, as in Figs. 16-13 d and e for a *pin-type puller*, in which an offset pin is the fastener. Pullers like these are hand operated and slow.

A popular type of *automatic puller* is shown in Fig. 16-13 C, for broach shanks like Fig. 16-13 f. When the machine ram moves all the way forward, the sleeve on the puller is pushed back by a stop. The keys are enabled to move outward into the groove in the sleeve and release the broach shank. Automatic pullers are especially desirable for machines that have broach elevators and automatic handling equipment. However, automatic pullers can be operated manually at a faster rate than other types.

Fixtures for broaching. Simple face plate fixtures as exemplified in Figs. 16-1, 16-2, and 16-5 are all that are needed for many broaching operations. For radial location of a part, pins, blocks, etc. may be added as illustrated in Fig. 16-4. If the face of a part is not likely to be square with the hole, it may be made to bear against a ring with a spherical seat. All details of a fixture or face plate that a broach might touch are left soft to avoid broach breakage.

Surface broaching fixtures are simple in comparison with those for many other machining processes but seldom as simple as those for internal broaching. Generally, the broaching forces tend to stabilize the part in the fixture, but broaching forces are large, and fixtures must be strong and rigid to give ample support to the parts. Surface broaching fixtures are shown in Figs. 16-3 and 16-7. Some fixtures are hand operated; others are actuated by the movement of the table and may be operated by air or hydraulic pressure.

Broaching Operations

Setup and operation. In broaching, skill is required to make the broach and other tools and to set up the equipment. After those things have been done, operation of the machine is mere routine. Because skill is built into broaching tools, they can be planned and designed to perform at maximum capacity and efficiency. When that is done, the equipment must be set up carefully and skilfully to assure that it will not be overloaded and will perform as intended. For internal broaching, the broach puller and outer support must be aligned with the center of the face plate or fixture to prevent the broach's drifting to one side. If the broach cuts heavily on one side the teeth on that side may be overloaded and damaged. For surface broaching, the broach inserts must be mounted carefully on their holder, and the holder on the ram, to be sure that each tooth is at its proper height and carries its share of the load but no more. Fixtures must be positioned and aligned carefully so as not to interfere with the broach but to be able to support the work securely. Rough workpieces must be checked for hardness, the correct amount of stock, and uniformity.

Speeds and stock removal. Broaching is done at comparatively low cutting speeds. Most broaching machines do not run at over 30

sfpm, and most operations are done at 12 to 24 sfpm. The return stroke is generally faster, up to 40 sfpm.

Each tooth of a broach is limited to removing a few thousandths of an inch of stock, and the total amount of stock that may be removed from a piece depends upon the length of the broach and the stroke of the machine. For most cases, $\frac{1}{4}$ in. is the most stock allowed, and generally $\frac{1}{8}$ in. is preferred. After a broach has been designed for a certain stock removal, the amount of stock must not be exceeded or the first teeth will be badly overloaded and damaged.

Material is best broached with a hardness between 10 and 35 C Rockwell. Harder material wears the broach teeth rapidly, and too soft material is difficult to cut cleanly. Hard spots and inclusions in workpiece material nick and damage broach teeth.

Planning for economical broaching. The cutting time in a broaching operation is short, and the loading and unloading time takes an appreciable part of the operation time. Thus, labor saving and motion economy are important considerations. Machines are arranged with multiple stations, broach handling is done automatically, and fixtures are made quick acting to save operation time. The extent of labor saving measures depends upon the quantity of production required. The cutting forces in broaching act in directions that tend to seat the parts and are favorable to simple and quick clamping. Clamping may be done by hand for moderately large quantities, but quick acting cams or levers are commonly provided. Air and hydraulic clamping devices are more intricate and expensive but warranted for very large quantities.

A broach must be carefully designed for large-quantity production. If a broach is too short, it will be overloaded and probably fail. On the other hand, too long a broach is unnecessarily expensive because the cost of a broach is largely determined by its length. What is more, too long a broach is inefficient because it requires excessively long cutting and return strokes.

Surfaces that run in the same direction on a workpiece, especially if they are contiguous, can generally be broached in one pass. As many such surfaces should be combined into one operation as feasible.

Although most broaching operations are completed in one pass, some are arranged for repeated cuts to simplify the design of the broach and save tooling expense where small or moderate quanti-

ties only are needed. The teeth of a gear or spline may be broached repetitiously. A comparatively simple broach is used to cut one or a few tooth spaces. After the first pass, the gear blank is indexed, and more of its teeth are cut. Successive passes are made until all the teeth are finished.

Where several similar parts are made in one plant in small or moderate quantities, arrangements may be made to broach them all economically with the same tooling. A machine tool manufacturer makes a variety of dogs of different sizes and shapes, but all have an integral key on one face. Broaching equipment has been designed to take any of the dogs, and all are broached on the key and face, although none alone is produced in sufficient quantity to justify the cost of broaching equipment.

Broaching compared with other operations. The work done by broaching can also be done in other ways. In this sense, broaching is competitive with boring and reaming on the lathe, drill press, etc., to produce round holes and with shaping and slotting for irregular internal surfaces. External surfaces may be finished by shaping, planing, facing, or milling as well as by surface broaching. Under certain conditions, broaching is preferable to other operations, but under other conditions broaching is either unfeasible or disadvantageous.

The main advantage of broaching is that it is fast. With properly applied broaches, fixtures, and machines, more pieces can be turned out per hour by broaching than by any other means. Little skill is required to perform a broaching operation after it is set up, and automation is easily arranged. The tool cost per piece is low. A broach does not need to be sharpened nor changed often and has a long life because speeds are low, each tooth makes contact only once and for a short time with each workpiece, and thus the broach does not have an opportunity to overheat. Roughing and finishing can be and often are done by one broach. Good finish and accuracy are obtainable over the life of a broach because roughing and finishing are done by separate teeth, and the finishing teeth are confined to removing only a little stock and are kept in good condition. Cutting fluid is readily applied where it is most effective because a broach tends to carry the fluid into the cut and does not throw it off like a rapidly revolving milling cutter.

Broaching has disadvantages as well as advantages. A broach is

essentially a form tool, and each one serves only a single purpose. Broaches are expensive to make and sharpen. Broaching machines must be capable of exerting large forces. Special fixtures are usually required for surface broaching. All this adds up to a relatively heavy investment for each broaching operation.

Special precautions in founding or forging to control variations in stock or extra operations to remove excess stock may have to be assumed before broaching to protect the broach. These extra considerations add to the over-all cost of manufacturing.

Some of the limitations of broaching are enough to make it impracticable for certain work. A surface cannot be broached if it has an obstruction across the path of broach travel. For instance, blind holes and pockets normally are not broached. Frail workpieces are not good subjects for broaching because they are not able to withstand the large forces imposed by the process without distorting or breaking. Large surfaces, especially long ones, cannot be broached with equipment of reasonable size because they require too much broach travel and excessively large broaches. Surfaces that run in the same general direction often can be broached at the same time, but surfaces that do not have such relative positions must be broached separately as a rule. For instance, a hole and a perpendicular face may be machined in one operation on a lathe or boring machine but require two passes in broaching. The lines left on a surface lie in the direction of broach travel. Thus for instance, broaching is not capable of producing a circular pattern in a hole if such a finish is required.

When broaching can be done, it is selected in preference to other processes if the amount it saves is more than able to pay for the cost of the equipment. The amount saved by broaching depends upon the saving per piece and the number of pieces required. The saving per piece may be relatively large in some cases, as, for instance, in machining small diameter but long holes with splines or keyways. In such cases, broaching may be economical for small or moderate quantities. For work that can be done readily in other ways, runs of 100,000 or more pieces are often necessary if broaching is to be considered.

Estimating broaching time. The cutting time in a broaching operation is the quotient of the length of stroke in inches divided by the cutting speed in inches per minute. For example, a stroke of

48 in. at 24 ft per minute is required for a broaching operation. The cutting time should be $48/(24 \times 12) = 1/6$ min = 10 sec. The return stroke may be calculated in the same way on the basis of the return speed of the broach but may not add to the operation time if unloading and loading can be done while the broach is being returned. Other elements of time in a broaching operation are starting and stopping time, commonly two seconds, and loading and unloading time which depends upon the nature of the work and fixture.

Questions

1. How are most modern broaching machines driven?
2. How is the size of a broaching machine designated?
3. Describe a broaching press and the kind of work done on it.
4. Describe a pull broaching machine and the kind of work done on it. What are its advantages and disadvantages?
5. Describe vertical pull up and pull down broaching machines and the work for which each is best suited.
6. Name and describe the common types of vertical surface broaching machines. Specify the purpose of each.
7. Sketch a typical pull broach and name its principal details.
8. What determines the amount of step between adjacent broach teeth?
9. How long should a broach be?
10. How much force is required to pull a broach?
11. What is a broach puller? Name and describe 3 common types.
12. Describe the types of fixtures used for internal broaching and for surface broaching.
13. Where is the skill required in broaching? Why?
14. What factors may have to be considered for economical broaching?
15. What are the advantages of broaching over other processes? The disadvantages?
16. When is broaching selected for a job?

References

Broaches and Broaching. New York: The Broaching Tool Institute, 1944.

Manual of Broaching. Detroit, Mich.: Detroit Broach Co., 1948.

Rylander, E. A., "The Elements of Broaching-Procedures, Tools, and Applications," *The Tool Engineer*, Vol. XXVII, No. 1, July, 1951.

Manufacturers' Catalogs:
American Broach and Machine Co., Ann Arbor, Mich.
Cincinnati Milling Machine Co., Cincinnati, Ohio.
Colonial Broach Co., Detroit, Mich.
Continental Tool Works, Division Ex-Cello-Co., Detroit, Mich.
Detroit Broach Co., Detroit, Mich.
Foote-Burt Co., Cleveland, Ohio.
Greenerd Arbor Press Co., Nashua, N. H.
Oilgear Co., Milwaukee, Wis.

Chapter 17

GRINDING MACHINES

Types of Grinding Machines

A GRINDING MACHINE UTILIZES AN ABRASIVE wheel revolving at high speed to cut material. Grinding is done on surfaces of almost all conceivable shapes and materials of all kinds. Grinding may be divided into two classes, according to its purpose. One class is called *precision grinding* and is concerned with producing good surface finishes and accurate dimensions. The other class is *non-precision grinding* to remove stock that cannot be taken off as well by other methods. In the latter case, the surface obtained is of secondary importance. Grinding machines may be made for one purpose or the other, and may be classified as precision or nonprecision grinders.

A precision grinding machine normally is designed to finish surfaces of a particular kind. *Cylindrical grinders* are precision grinders for round surfaces. Some are intended for external surfaces, others for internal surfaces. *Surface grinders* are for plane surfaces. In addition, specialized precision grinders are made for particular applications. Among these are gear grinders (described in Chapter 20), thread grinders, cam grinders, crankshaft grinders, camshaft grinders, and way grinders. Also, a number of types of grinders are designed for making and sharpening tools.

Cylindrical Grinders

Toolpost grinders. A toolpost grinder is a grinding wheel head that may be attached to the compound rest of a lathe for cylindrical grinding or to the tool head of a planer or shaper for surface grind-

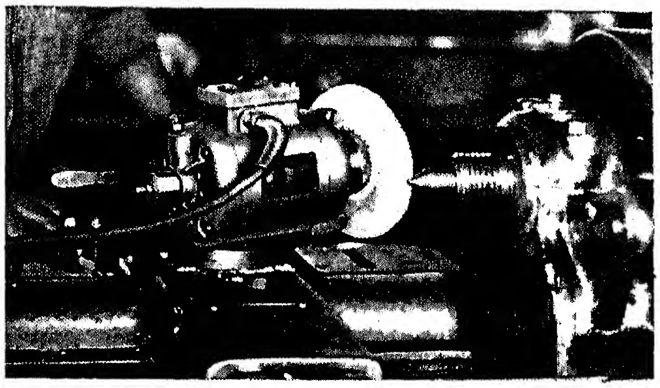


Fig. 17-1. A tool post grinder grinding a center on a lathe. (Courtesy The United States Electrical Tool Co.)

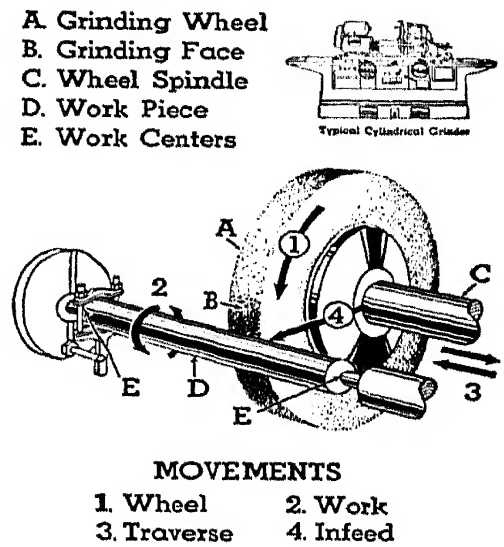


Fig. 17-2. The movements of an external cylindrical centertype grinding machine. (Courtesy The Carborundum Co.)

ing. A center is being ground on a lathe in Fig. 17-1 by a toolpost grinder traversed by the compound rest swiveled at an angle. This equipment is capable of light work where machines specifically for grinding are not readily available. Toolpost grinders are made in various sizes with motors from ¼ to 10 hp. Some models run at high speeds to drive small wheels for internal grinding; others take larger wheels for external grinding.

Plain centertype grinders. A workpiece is usually held and rotated between dead centers on a *plain cylindrical centertype grinder* as depicted in Fig. 17-2. The centers in the headstock and tailstock do not revolve because the most rigid work support and accuracy can be obtained in that way. A balanced dog on the workpiece is engaged by a driver, usually a pin, on the face plate that revolves around the dead spindle of the headstock. The tailstock center is retracted to insert a workpiece between centers. That is usually done by hand, but some grinders have air- or hydraulic-operated tailstocks for quantity production.

The headstock and tailstock of a plain grinder are mounted on an upper or swivel table and can be positioned along the table to suit the work. The table is flat on some plain grinders but has a triangular section on others, like the one in Fig. 17-3. The triangular table is rigid, and cutting fluid runs off the slanting top to the basin in the bed. The upper table can be swiveled and clamped in position on a lower table to provide adjustment for grinding straight

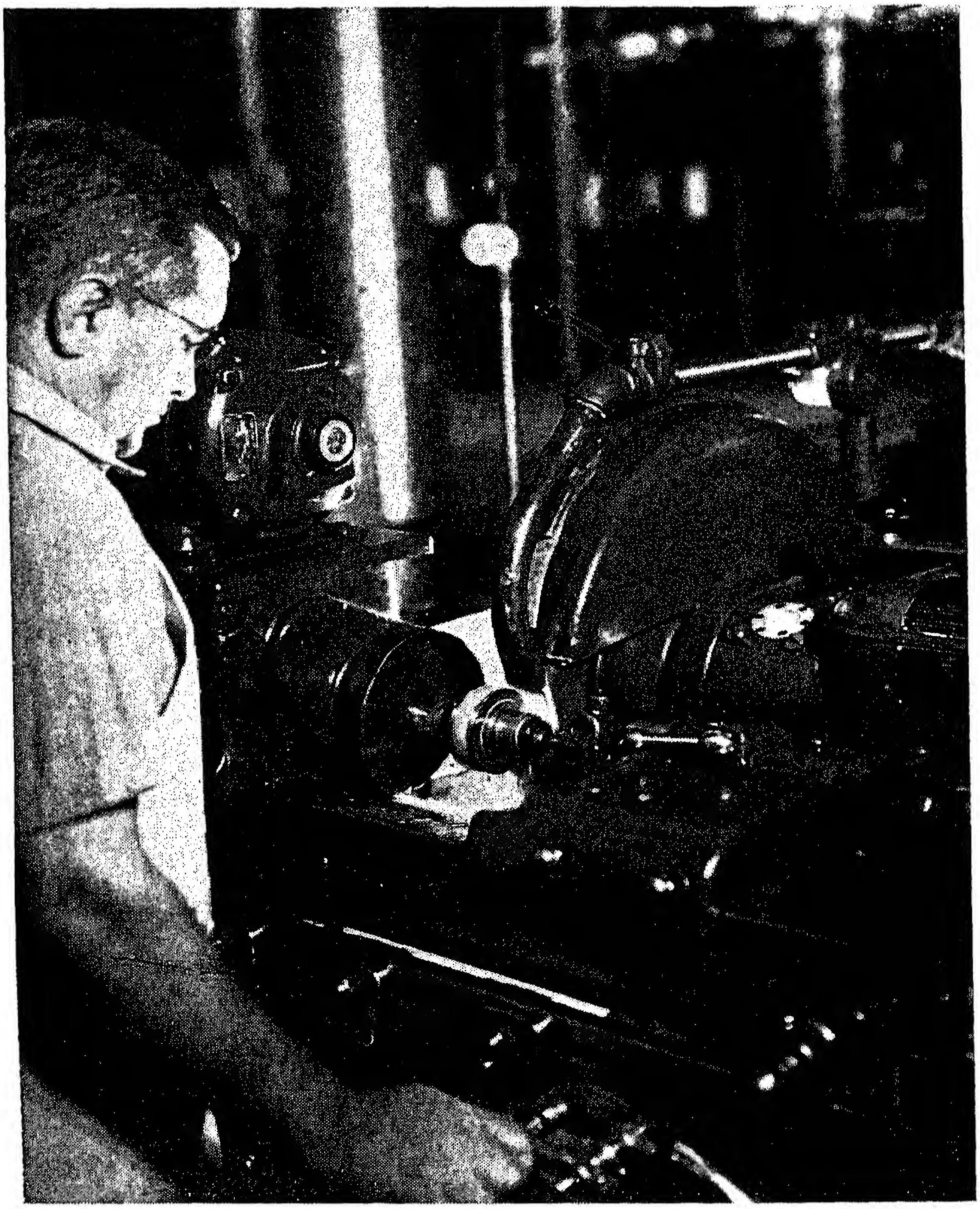


Fig. 17-3. A plain cylindrical centertype grinding machine in operation. (Courtesy Norton Co.)

or tapered work as desired. The amount of taper that can be ground is normally limited to less than 10° because the table bumps the wheelhead if swiveled too far.

The lower table slides on ways and provides longitudinal traverse for the workpiece. It can be moved manually through a handwheel on the front of the bed and also by power. Most modern machines

have hydraulic table drives that provide infinite increments of adjustment over wide ranges of traverse speeds, smooth traverse, and closely controllable table reversal. Dogs on the front of the sliding table are set to reverse the power traverse at desired points.

The grinding wheel revolves in the direction shown in Fig. 17-2 on a heavy spindle running in close fitting bearings to prevent flutter. The grinding wheel face and workpiece surface travel in opposite directions at their line of contact. The grinding force is directed downward for stability. The spindle is driven by a motor, usually on the back of the wheelhead. Most grinders have belt drives on their workheads and wheelheads to avoid vibration which would show up on the ground surfaces as chatter marks.

The wheelhead on most plain grinders is mounted on ways on the bed and carries the wheel to the work. The movement is called infeed, as designated in Fig. 71-2, and is obtained from an accurate screw and nut controlled by a handwheel, grasped by the operator's right hand in Fig. 17-3. A geared drive between the handwheel and screw makes adjustments possible to as close as 0.0001 in. In addition, many machines have a hydraulic infeed mechanism that feeds the wheel in to a definite position to enable it to grind the work to size, and then retracts the wheel to allow the work to be unloaded and loaded for each cycle of an operation.

The base or bed of a plain grinder is a massive boxlike structure with heavy ribs, designed to absorb vibrations and resist deflections. It contains a storage compartment for the cutting fluid, which is pumped to and discharged from a nozzle over the grinding zone. A copious supply of fluid is necessary for rapid, accurate, and efficient grinding. A heavy guard covers most of the grinding wheel for safety.

Roll grinders. Cylindrical centertype grinders capable of swinging diameters in excess of 20 in. and equipped specifically for grinding rolls for rolling mills are called roll grinders. They resemble plain grinders in their basic features. Some roll grinders have only one table but a set-over tailstock, like that on a lathe, to provide adjustments for grinding straight or tapered work. Otherwise the moderate-size roll grinders have all the movements already described for plain grinders.

Equipment ordinarily needed for roll grinding includes journal rests and a cambering attachment. Heavy rolls are usually revolved

on their journals while being ground. In that way, the outside of the roll is ground concentric with the journals on which it runs in service. The journals of a roll may be ground first, with the roll between centers. A large roll on journal rests is shown in Fig. 17-4.

Rolls that cold work metal are subject to large forces that spring even the most massive. Rolls that work on hot metal become hot themselves and expand more at the center than at the ends. Consequently to roll metal of uniform thickness, rolls have to be larger

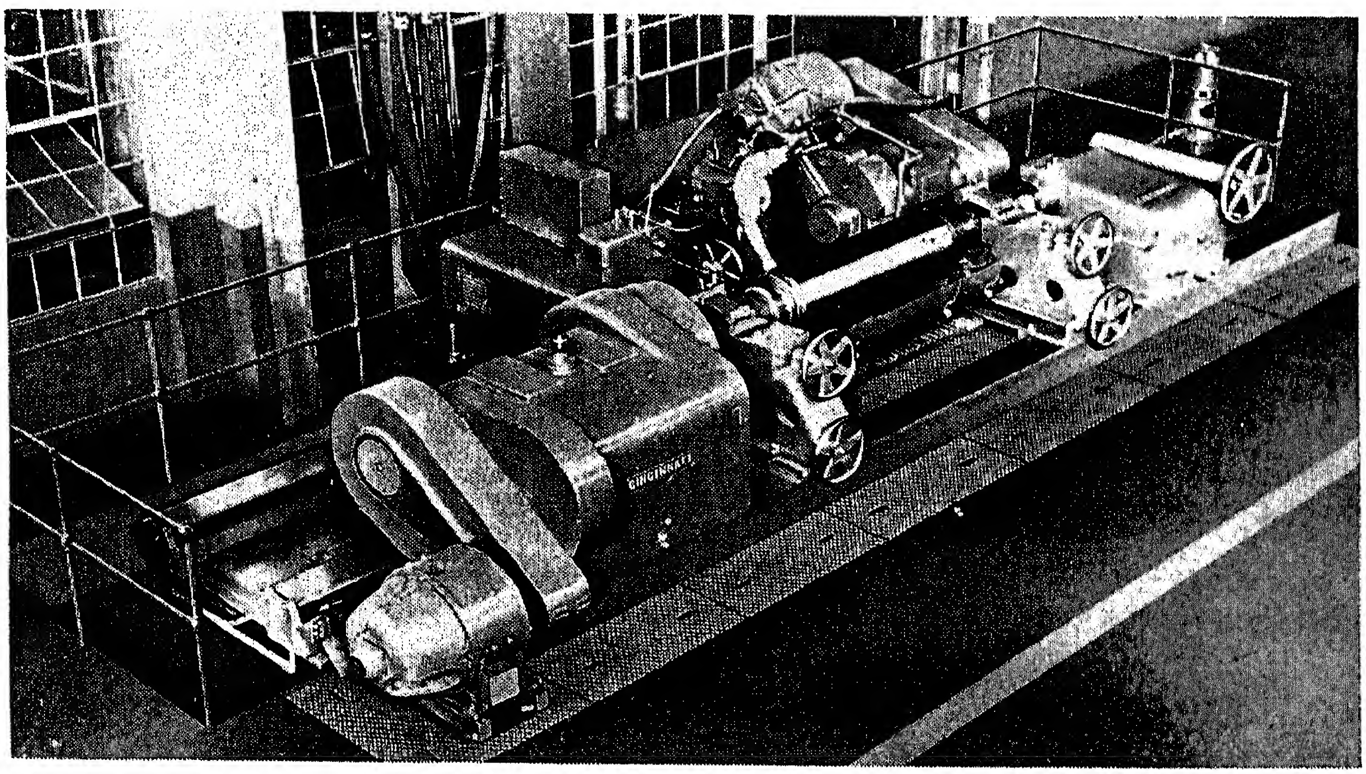


Fig. 17-4. A 36 in. by 168 in. traveling head roll grinder. (Courtesy Cincinnati Grinders, Inc.)

or smaller in the middle than at the ends. Such rolls are said to be crowned or cambered, the amount depending upon the specific application. The curvature must be accurate and reproducible whenever rolls are ground or reground. Various kinds of cambering attachments have been applied to roll grinders to enable them to put the desired contours on rolls. A common form has a cam that moves in unison with the worktable. The cam is behind the wheelhead and causes the grinding wheel to move in or out as the roll is traversed past the wheel.

Roll grinders for diameters over about 36 in. have a unique design. One is shown in Fig. 17-4. The heaviest rolls are not

traversed. The headstock, footstock, and journal rests are mounted on the bed. The tailstock is of the set-over type. The grinding wheelhead rides on a carriage traversed along the work on ways on the bed. The wheelhead is fed into the work. The controls are next to the operator stationed on the carriage where he can observe the grinding action.

Universal centertype grinders. A universal centertype grinder, as illustrated in Fig. 17-5, has all the units and movements of a plain

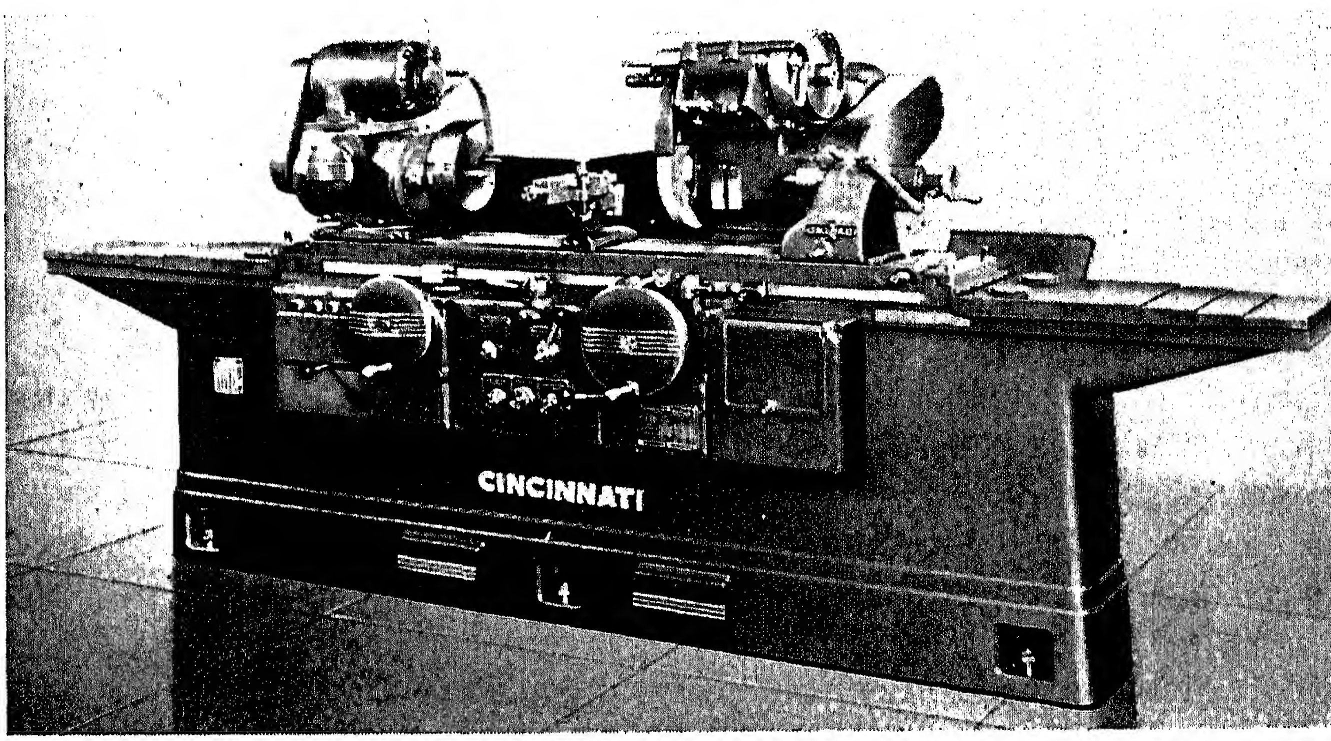


Fig. 17-5. A 12 in. by 36 in. universal centertype hydraulic grinding machine. (Courtesy Cincinnati Grinders, Inc.)

grinder, but in addition its headstock can be swiveled in a horizontal plain, its headstock spindle may be used alive or dead, and its wheelhead and slide can be swiveled. These features make a universal grinder more versatile but less capable of grinding rapidly than a plain grinder.

The headstock spindle of a universal grinder can be locked to keep it from turning for grinding between centers and can also be released so that it can be rotated by the face plate. As a live spindle, it may carry and rotate a chuck, face plate, or fixture for holding work. The headstock is mounted on a base on a flat table and can be swiveled in a horizontal plane through 360°. Thus, chuck work can be positioned for grinding tapers of all angles with the face of

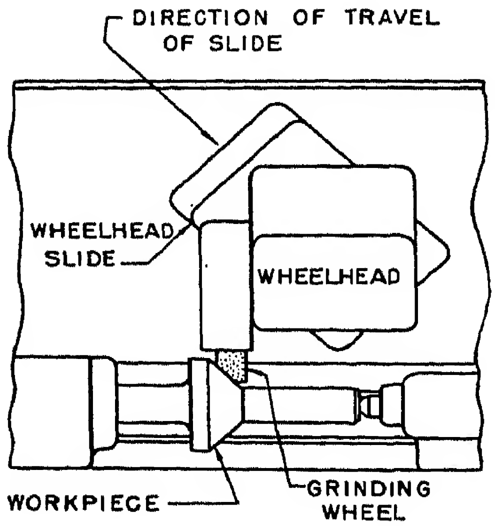


Fig. 17-6. A diagram of a universal centertype grinder arranged for grinding a steep taper.

the wheel. If the headstock is swiveled at right angles to its normal position, a flat surface can be ground on chuck work by rotating the workpiece and traversing it past the grinding wheel.

The wheelhead slide of a universal grinder can be swiveled in a horizontal plane, and the angle of swivel read from a graduated ring. The wheelhead on top of the slide can then be swiveled back to its normal or any desired position. The wheel is fed at the angle to which the slide is swiveled. Thus, any degree of taper can be ground on work between centers. A universal grinder with its wheelhead slide swiveled at an angle and the wheelhead returned to its normal position to grind a steep taper on a workpiece is depicted in Fig. 17-6. The face of the wheel is trued to the desired angle by traversing the wheelhead slide. Tapers that are not steep can also be ground by swiveling the table and using the wheelhead for infeed only, as is done on a plain grinder. If both table and wheelhead are swiveled, a gradual taper and a steep taper can be ground on a workpiece in one setup.

Centertype grinder attachments and accessories. A number of devices are available to increase the usefulness and facilitate the operation of cylindrical centertype grinders.

Backrests or *steady rests* are attachments to support long and slender workpieces that tend to deflect between centers when subjected to grinding forces. A backrest on a cylindrical grinder serves the same purpose as a center rest on a lathe. One type of backrest, mounted on the table of the grinder in Fig. 17-5, has two adjustable wooden shoes that bear on the side and bottom of a workpiece. The shoes are adjusted separately by hand to make contact with the workpiece surface. Another type of backrest is known as the spring type and has steel shoes that are held against the work by springs and move in as the work diameter is reduced by grinding. Backrests are spaced along a grinding machine table at intervals equal to about 6 to 10 times the diameter of the workpiece.

When a workpiece is not traversed, but instead the grinding

wheel is plunged into the work, the wheel is often given an axial movement of $\frac{1}{16}$ to $\frac{1}{4}$ in. to help it wipe up the surface. That is called *reciprocation* of the grinding wheel. The simplest reciprocating device is a long lever pivoted from the wheelhead with one end engaged with a collar on the grinding wheel spindle, and the other end manipulated by the operator. For production work, the spindle is reciprocated continuously by a hydraulic or mechanical device.

A universal grinder may be arranged for internal grinding by the addition of an auxiliary wheelhead to revolve small wheels at high speeds. The internal grinding attachment on the universal grinder of Fig. 17-5 is hinged on the front of the main wheelhead and is swung down to a position above the table for use. It serves for internal grinding of pieces held in a chuck, face plate, or fixture on the live spindle of the headstock.

Attachments for balancing and truing grinding wheels are necessary for operating centertype grinders properly, as well as for other types of grinders, and will be described later.

Sizes and uses of cylindrical centertype grinders. The size of a cylindrical centertype grinder usually is designated by the diameter in inches and the length in inches of the largest workpiece the machine can nominally accommodate between centers. Thus, a 12 in. by 36 in. centertype grinder can swing a workpiece 12 in. in diameter over the table and grind it with a new wheel. A workpiece up to 36 in. long may be mounted between centers. Some manufacturers designate the sizes of their grinders by numbers, such as 1, 2, etc., but specify in catalog descriptions the maximum diameter and length of work taken by each size of machine.

Plain grinders range in size from about 3 by 12 in. to 28 by 192 in. Some roll grinders are capable of taking work 60 in. in diameter by 24 ft. long. Universal grinders commonly are made in sizes from 10 by 24 in. to 18 by 72 in.

Cylindrical centertype grinders are used for grinding straight and tapered round pieces, round parts with curved profiles, fillets, shoulders, and faces. Plain grinders are designed to remove stock as rapidly as feasible and to produce accurate ground surfaces as quickly as possible on pieces supported and rotated between centers. They are production-type machines. A typical 10 by 18 in. plain grinder has a 15 hp wheelhead drive motor, a 1 hp headstock motor, and a $1\frac{1}{2}$ hp hydraulic unit drive motor. Universal grinders

can grind surfaces such as steep tapers and holes not accommodated on plain grinders but at the sacrifice of rigidity and rapidity of operation. They are found in toolrooms and jobbing shops. A 12 by 36 in. universal grinder has a 3 hp wheelhead motor, a ¾ hp headstock motor, and a 1 hp hydraulic drive motor.

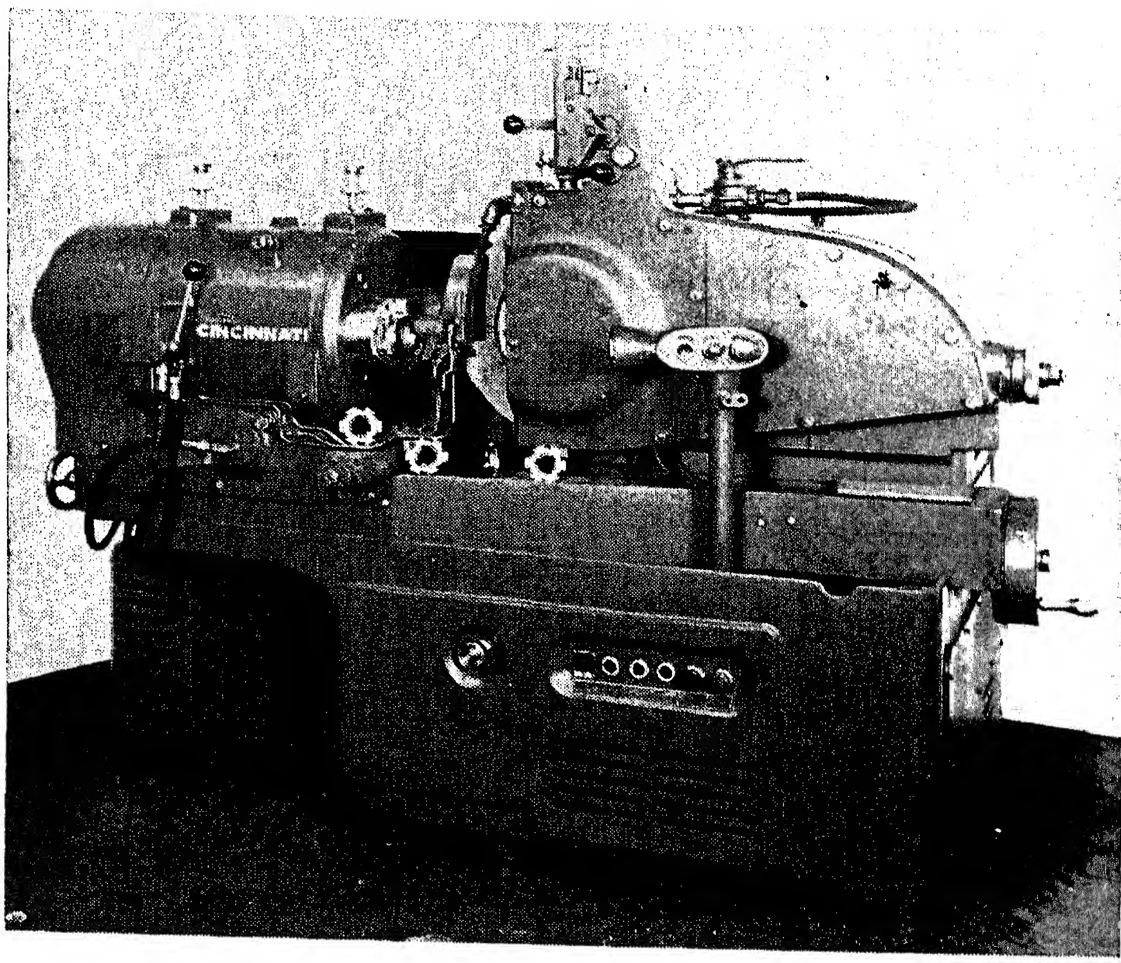


Fig. 17-7. A 2 spindle chucking grinder. (Courtesy Cincinnati Grinders, Inc.)

Chucking grinders. Chucking grinders are designed for grinding small and medium diameter short parts in large quantities. Typical applications are the grinding of tapered roller bearing cone races, valve tappets, and small bevel gear shoulders and stems. The work is held in chucks, collets, or fixtures.

The cycle of a chucking grinder is generally automatic. The operator loads a piece and presses a button. The work is clamped and rotated. The grinding wheel advances rapidly to the work, then proceeds at feed rate to grind the work to size, dwells, and retracts to starting position. The work rotation stops, and the workpiece is

released from the chuck. Quite often, chucking grinders are equipped with automatic loading and unloading attachments. In those cases, all the operator has to do after the machine has been set up is to keep a magazine filled with pieces.

Most chucking grinders hold one piece at a time, but the machine in Fig. 17-7 has two work spindles. While a part in one spindle is being ground, the other spindle is unloaded and loaded. The head automatically shifts from one grinding position to the other. Grinding is practically continuous; the only nonproductive time is three seconds for indexing the workhead.

Centerless grinders. An *external cylindrical centerless grinding machine* revolves a workpiece on top of a workrest blade between two abrasive wheels as shown in Fig. 17-8. The blade is made of hard wear resistant material, and its top slants downward toward the regulating or driving wheel. Usually the workrest blade is raised so that the work center is higher than the wheel centers because that helps make the workpiece round. If the work center is at the same height as the wheel centers, the piece is not likely to be round. Long pieces may be ground below the wheel centers to keep down whipping and chattering.

The grinding wheel revolves at high speed, up to 6500 sfpm in

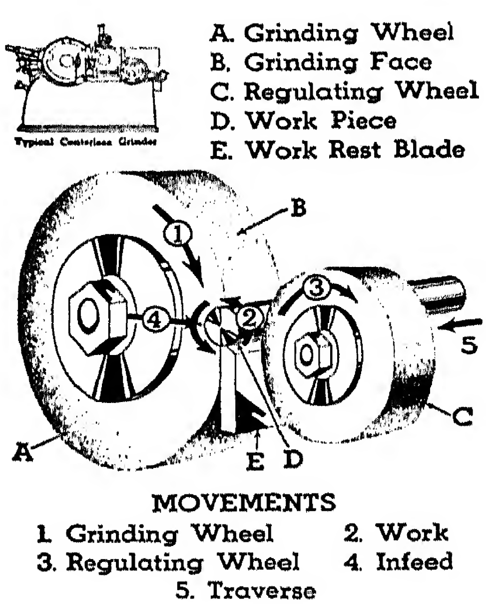


Fig. 17-8. The action of an external cylindrical centerless grinding machine. (Courtesy The Carborundum Co.)

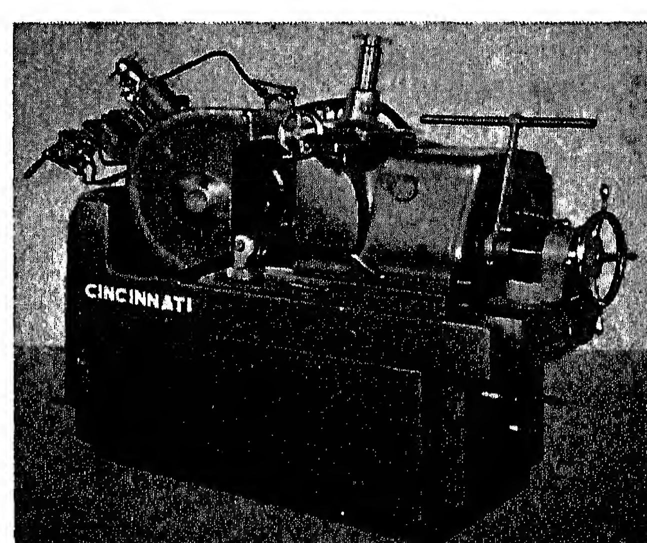


Fig. 17-9. A No. 2 centerless grinder arranged for infeed grinding. (Courtesy Cincinnati Grinders, Inc.)

most cases. Its face travels downward in contact with the work surface, moving in the same direction. The grinding wheel tends to turn the workpiece fast but also presses the piece against the regulating wheel on the other side. The workpiece has a greater affinity for the regulating wheel, which is usually a rubber bonded abrasive wheel, and normally adheres to the surface speed of the regulating wheel, which is from 50 to 200 sfpm. The regulating wheel surface in contact with the work travels upward.

A typical medium-size centerless grinder is shown in Fig. 17-9. The grinding wheel is carried on the end of a spindle that revolves in bearings in a wheelhead integral with the base. The spindle is driven at constant speed through belts by a 15 hp motor. The grinding wheel is trued by a hydraulically traversed attachment on the heavy wheelguard. These units are on the left end of the machine of Fig. 17-9.

The workrest blade is fastened in a workrest, is mounted on a lower slide on the base, and can be moved toward or away from the grinding wheel. A clamping lever on the front of the lower slide provides means to fasten the slide in any desired position. An upper slide can be moved on the lower slide and clamped as desired. The upper slide carries the regulating wheel housing on a swivel so that the regulating wheel can be tilted a few degrees about a horizontal axis at right angles to the axis of the wheel. When the regulating wheel is tilted, it causes the work to move axially between the wheels. A screw-driven truing device on top of the regulating wheel housing is swiveled to true a shape on the regulating wheel so that the wheel makes full lengthwise contact with the workpiece.

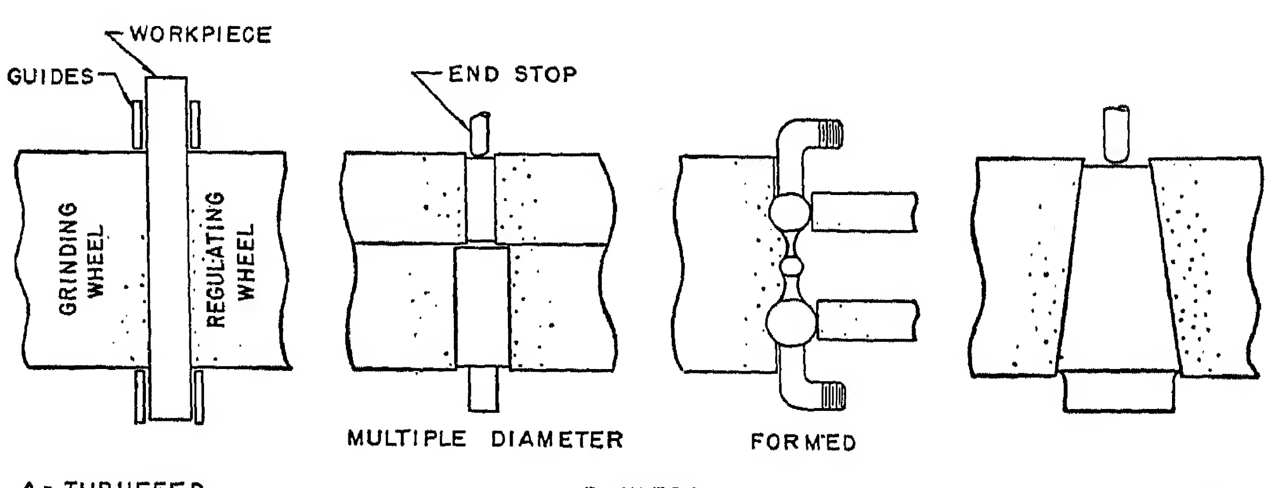


Fig. 17-10. Centerless grinding operations.

A nut on the upper slide of the centerless grinder of Fig. 17-9 is engaged with a screw attached to the bracket on top of the right end of the bed. The handwheel shown and a graduated dial are carried on the end of the screw. If the screw is turned while the upper slide is free on the lower slide and the lower slide is clamped to the bed, the regulating wheel is moved toward or away from the workrest blade. In that way the position of a workpiece on the blade and against the regulating wheel is established. Then if the screw is turned while the lower slide is free on the bed and the two slides are clamped together, the workblade and regulating wheel are moved together toward or away from the grinding wheel. That positions the workpiece with respect to the grinding wheel and determines the size to which the piece is ground.

A 1½ hp motor drives the regulating wheel and an oil pump. The speed of the regulating wheel is changed by shifting gears by means of the levers on the lower right end of the bed.

Thrufeed centerless grinding is done by passing the workpiece completely through the space between the grinding and regulating wheel as indicated in Fig. 17-10 A. The work is supported by a long workrest blade and adjustable guides on both sides of the wheels. The rate of traverse depends upon the angle of inclination and speed of the regulating wheel, as expressed by the relation

$$F = \pi DN \sin \alpha.$$

F is the rate of feed of the work in inches per minute., D is the diameter of the regulating wheel in inches, N is the speed of the regulating wheel in rpm, and α is the angle to which the regulating wheel is tilted.

The angle of tilt of the regulating wheel may be from 0° to 8°. The amount of tilt and the speed of the regulating wheel determine the amount of time available in a pass to remove stock, make the workpiece round, and provide a good finish. A slow feed is necessary to remove relatively large amounts of stock and produce accuracy and good surface finish. Most thrufeed work is ground in 2 passes with a total stock removal of 0.010 to 0.015 in.

The grinding wheel wears smaller as successive pieces are ground. The slides are moved in at intervals by means of the handwheel and dial on the end of the bed to keep the machine grinding to the desired size.

Infeed centerless grinding is slower than thrufeed grinding but is necessary for a workpiece with a shoulder, head, or obstruction that prevents its passing completely through the throat between the wheels. Two examples of infeed grinding are given in Fig. 17-10 B. The workrest and regulating wheel are withdrawn from the grinding wheel to receive a workpiece. They are then moved towards the grinding wheel to grind the work to size. The movements may be performed manually by means of the lever with the long bar handle on the end of the upper slide housing. That lever turns the nut on the upper slide engaged with the infeed screw. For high production, the slide may be moved in and out automatically by a mechanical or hydraulic attachment.

The regulating wheel is tilted at only a small angle for infeed grinding, just enough to hold the workpiece against the end stop. For pieces that can be pushed out between the wheels, the end stop is arranged as an ejector to kick out the work after it has been withdrawn from the grinding wheel. Its action may be manual or automatic. Automatic loading attachments are used, synchronized with the infeed attachment, for high production.

The grinding wheel must be as wide as the surface ground for true infeed grinding. Sometimes several wheels are mounted side by side. The wheels may be trued in steps or special forms as indicated in Fig. 17-10 B. These forms are obtained by guiding the truing attachments with special cam bars.

Endfeed centerless grinding is for tapered work, as shown in Fig. 17-10 C. Either the grinding wheel or regulating wheel or both are trued on a taper. The work is fed in from the front and advances as it is ground until it reaches the end stop.

Centerless grinder attachments. The *roller infeed workrest* of Fig. 17-11 supports long pieces ground on one end only.

Long bar attachments are used for thrufeed grinding long rods, tubes, and round bars. A typical long bar attachment consists of brackets extending from both ends of the workrest and carrying rollers that support the work before it enters and after it leaves the space between the wheels.

Automatic work-handling attachments for large-quantity production include hoppers and magazines from which pieces are released one after another for both thrufeed and infeed grinding. Where a number of passes is needed for parts, several centerless grinders may

be placed in a line and interconnected by conveyors so that pieces are passed from one machine to another without manual handling between the machines.

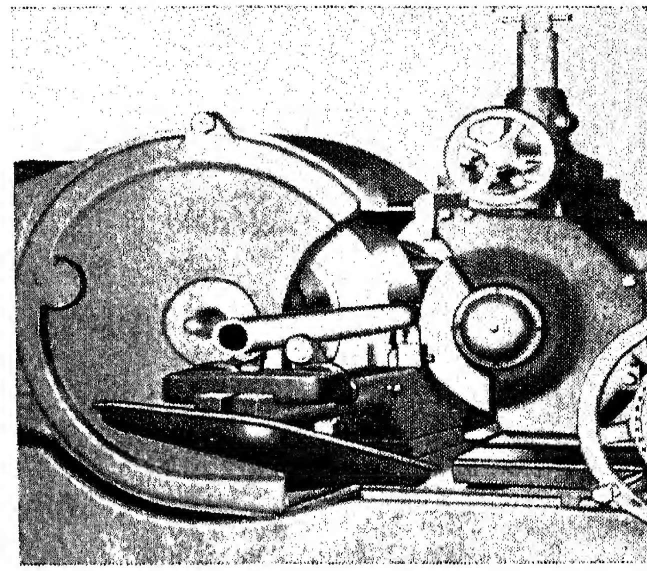


Fig. 17-11. A roller infeed workrest with outboard support. (Courtesy Cincinnati Grinders, Inc.)

Comparison of centertype and centerless grinding. Centerless grinding is a relatively new process. Since its inception in the early part of this century, it has replaced centertype grinding for most large quantity production. As a rule, the time to set up a centerless grinder is more than that needed to set up a centertype grinder, but the difference is not large for many simple parts. Also much can be done to minimize centerless setup time by scheduling similar parts in successive lots over the same machine. Parts with several diameters or curved profiles usually require special equipment that is expensive and takes a long time for setup. Therefore, irregular parts are not centerless ground unless produced in quite large quantities.

Centerless grinding is a faster operation than centertype grinding after a setup has been made. For simple parts, the saving in grinding time for only a few pieces may make up for the longer setup time for centerless grinding. Centerless grinding has been found profitable in many places for lots less than 100 pieces, sometimes for a dozen or less. However, as a rule, centertype grinding is preferable where the work is varied, irregular in shape, or large in size, especially in small quantities.

Centerless grinding removes stock rapidly and produces finished surfaces in the shortest time possible because:

1. It is almost continuous, especially for thrufeed grinding, with a minimum of machine time lost for loading and unloading.
2. The work is fully supported by the workrest blade and regulating wheel and can be subjected to cuts as heavy as it will take without overheating. Plunge cuts can often be made over the entire length of a workpiece.

3. No axial thrust is present, as it is on work between centers. Long thin pieces are not so likely to be distorted.
4. The action of centerless grinding is such that each workpiece is cleaned up with the removal of the least possible amount of stock. Errors of centering are eliminated.
5. Large grinding wheels can be used and wheel wear is minimized. Therefore, a minimum amount of adjustment is needed.
6. Adjustments for sizes are made directly on a centerless grinder, and that contributes to accuracy. If the regulating wheel and workrest blade are moved 0.001 in. toward the grinding wheel, the work diameter is reduced 0.001 in. If the grinding wheel on a centertype grinder is fed in 0.001 in., the work diameter is reduced 0.002 in.
7. A low order of skill is needed to attend to centerless grinding most of the time.

Conventional internal grinders. Conventional internal cylindrical grinders rotate the workpiece around the axis of the hole ground. The *plain internal grinder* of Fig. 17-12 is a typical semiautomatic machine designed to handle a variety of work of the same general character efficiently on short and moderately long production runs. The work is held in a chuck, clamped to a face plate, or located in a fixture on the spindle inside the guard on the workhead on the left of the base. The workhead can be swiveled to grind a straight hole or tapers up to 45° included angle. A dial indicator is provided for angular settings of the workhead. A 1½ hp motor drives the work spindle, and four spindle speeds from 133 to 400 rpm are available.

The high speed grinding wheelhead is powered by a 5 hp motor and is carried on a cross slide that is moved through a screw and nut by the infeed handwheel on the front. The cross slide is carried on a table that traverses lengthwise on the bed and moves the wheel to and from and through the hole in the workpiece. A handwheel on the bed can be made to move the table by hand. For grinding, the table is actuated by a low pressure, locked feed hydraulic system with rapid traverse speeds from 0 to 35 fpm and grinding speeds from 0 to 15 fpm. A 1 hp motor drives the hydraulic pump. Dogs on the front reverse the table. A short stroke is set

to reciprocate the wheel over the length of hole being ground. One of the dogs is raised by a foot pedal to withdraw the wheel from the hole.

The machine of Fig. 17-12 has an automatic as well as manual cross-feed. The automatic cross-feed can be adjusted for roughing to feed the wheel an amount equivalent to 0.0001 in to 0.001 in. on

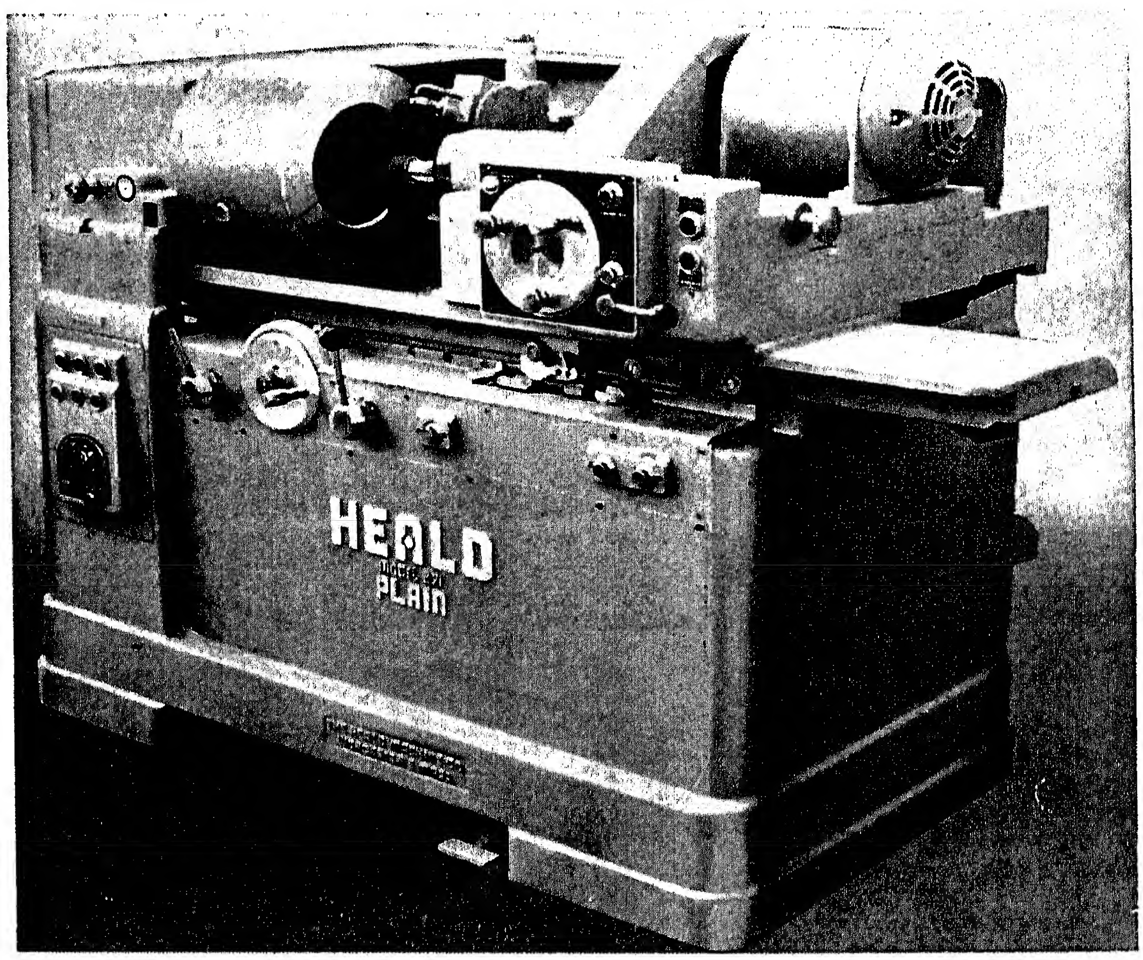


Fig. 17-12. A plain internal grinding machine for holes up to 7½ in. diameter by 5 in. long. (Courtesy The Heald Machine Co.)

the diameter of the work for each stroke of the wheel. The cross-feed for finishing is 0.0001 in. on the diameter per stroke.

A truing attachment is mounted behind the table and between the workhead and wheelhead. It carries a diamond on a pivoted arm that can be dropped in place for truing the wheel or swung out of the way.

A *universal internal grinder* is basically the same as a plain internal grinder except that the workhead is mounted on a cross slide, as is the wheelhead, and can be swiveled through a 90° angle. The

universal internal grinder is capable of grinding straight or tapered holes and external round surfaces, and flat, concave, or convex surfaces, either separately or together on one workpiece.

A *chuck-type internal grinder* has its workhead mounted on a reciprocating table and its wheelhead on a cross slide on the bed. The work is moved to and from and across the grinding wheel. This construction is simple and compact, gives the fullest support to the wheelhead, and permits very high wheel speeds.

A *gap internal grinding machine* has an open section or gap in the bed under the front of the work spindle to accommodate large diameter workpieces. A *duplex internal grinder* has a workhead between two reciprocating wheelheads so that two opposite holes in a workpiece can be ground in accurate alignment.

The various types of conventional internal grinders are made in many sizes from light machines for small work to heavy machines for large workpieces, 5 feet and more in diameter. The sizes of some makes are designated by the diameters of work that can be swung.

Internal grinder attachments and operations. Among the many kinds of work-holding devices used on internal grinders are independent four jaw chucks, collet chucks, face plates, and fixtures. A widely used chuck for internal grinding has jaws that slide on tapered blocks in the body of the chuck. The jaws close together to grip the work as they are pulled back equally by an air cylinder and draw rod. A *sliding jaw chuck* is illustrated in Fig. 17-13. The contact surfaces of the jaws are commonly ground in place on the machine while held under gripping pressure against a spider. A concentricity of 0.001 in total indicator reading is possible with this type of chuck. Another type of chuck is called a *diaphragm chuck* because its jaws are mounted on a steel diaphragm that is sprung outward to open the jaws. The jaws are ground in place, and concentricity as close as 0.0002 in. total indicator reading can be held on a diaphragm chuck.

Accurate sizing is necessary in internal grinding. For one or a few pieces, an operator can grind a hole to size by use of the cross-feed handwheel and dial. However, this is a slow process because several measurements must be made on each hole and care is necessary. The small wheels needed to get into holes wear rapidly and slender spindles deflect appreciably. One form of sizing device

that shows an operator the size of a hole being ground is depicted in Fig. 17-13. A finger rides on the hole surface and is connected to the dial indicator by a linkage. The arm that carries the finger can be swung out of the way for unloading and loading the work.

A *Size-matic* internal grinder is arranged to go through a routine to size each piece in production. The grinding wheel reciprocates through the hole and is fed automatically almost to size while it is roughing. The wheel is withdrawn, trued, and returned to the work. The truing diamond is set to correspond to the finished surface desired on the workpiece. After the wheel has been trued, it is fed in a predetermined amount that produces the desired size in the work.

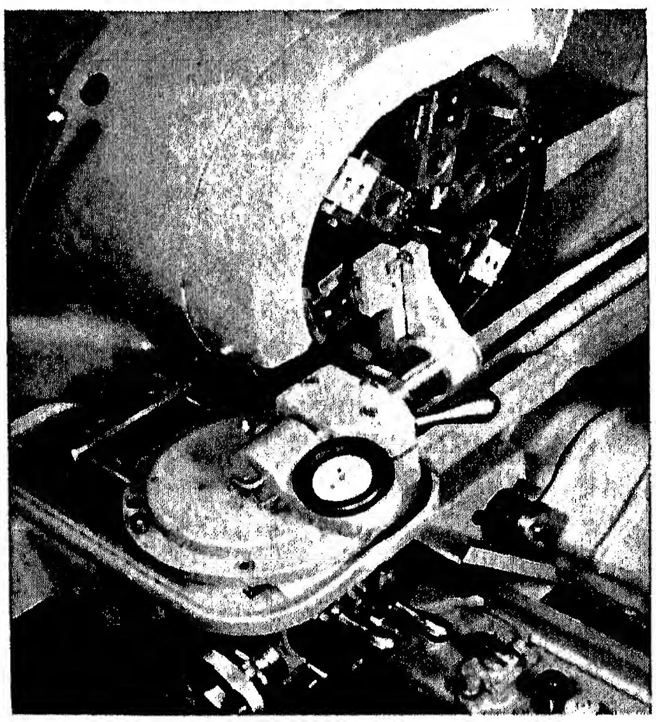


Fig. 17-13. A manually operated sizing device on an internal grinder. The workpiece is a gear located on its pitch line by pins in a sliding jaw chuck. (Courtesy Bryant Chucking Grinder Co.)

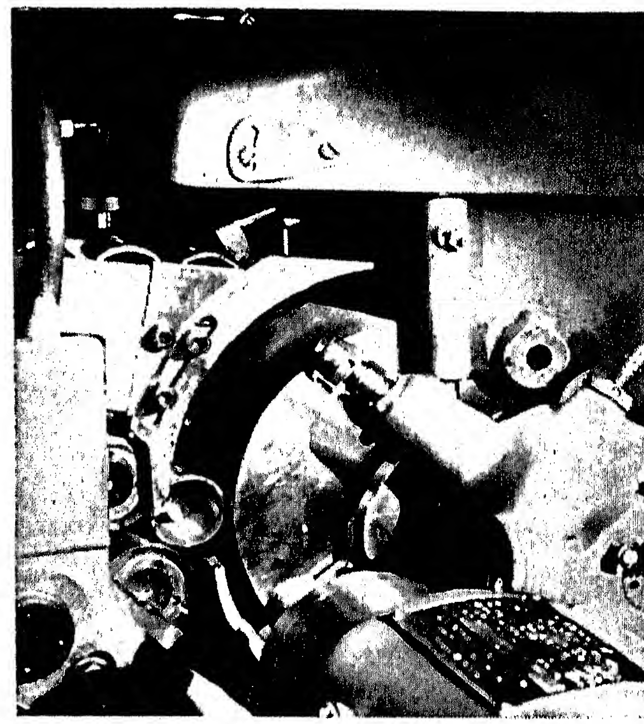


Fig. 17-14. A view of a centerless internal grinding machine. (Courtesy The Heald Machine Co.)

A *Gage-matic* internal grinder has a mechanism that tries to insert a round plug gage in the back of the hole each time the reciprocating wheel leaves the front of the hole being ground. When the first part of the plug enters, the wheel is trued for finishing. When the whole plug enters, the correct size is indicated, the wheel is withdrawn, and the machine stops.

For some purposes, a hole and face on a workpiece must be ground in the same operation to assure their being square. An *internal and face grinder* has two wheelheads side by side and two wheels, a small one for the hole and a large one for the face. The wheel slide is moved crosswise to bring each wheel in turn into position for grinding.

An internal grinder may be equipped with micrometer stops to position grinding wheels to grind the desired depths in holes or for face grinding.

Some internal grinders can be equipped with cams to guide the grinding wheels along angular, stepped, or curved paths. Such an arrangement provides means for grinding tapered, stepped, and other holes that are not straight at the fastest possible rate in quantities.

Centerless internal grinder. The centerless internal grinder revolves the workpiece between three rolls as shown in Fig. 17-14. The workpiece rests on a supporting roll, is held down by a pressure roll, and is driven by a large regulating roll. The grinding wheel is advanced to, traversed through, fed into, and retracted from the workpiece in the conventional manner described for other internal grinders.

The bore of the workpiece is ground uniformly in relation to the outside surface on which the piece rolls, and a high degree of concentricity is attainable between inside and outside surfaces. The manner of holding the work also lends itself well to automatic unloading and loading of the work. A loading blade ejects each workpiece after it is ground and helps guide into grinding position each new piece released from the chute above the wheels.

The centerless internal grinder can be arranged for either Sizematic or Gage-matic grinding and will handle straight, tapered, continuous, interrupted, open end, or blind holes in all parts having finished round outside surfaces.

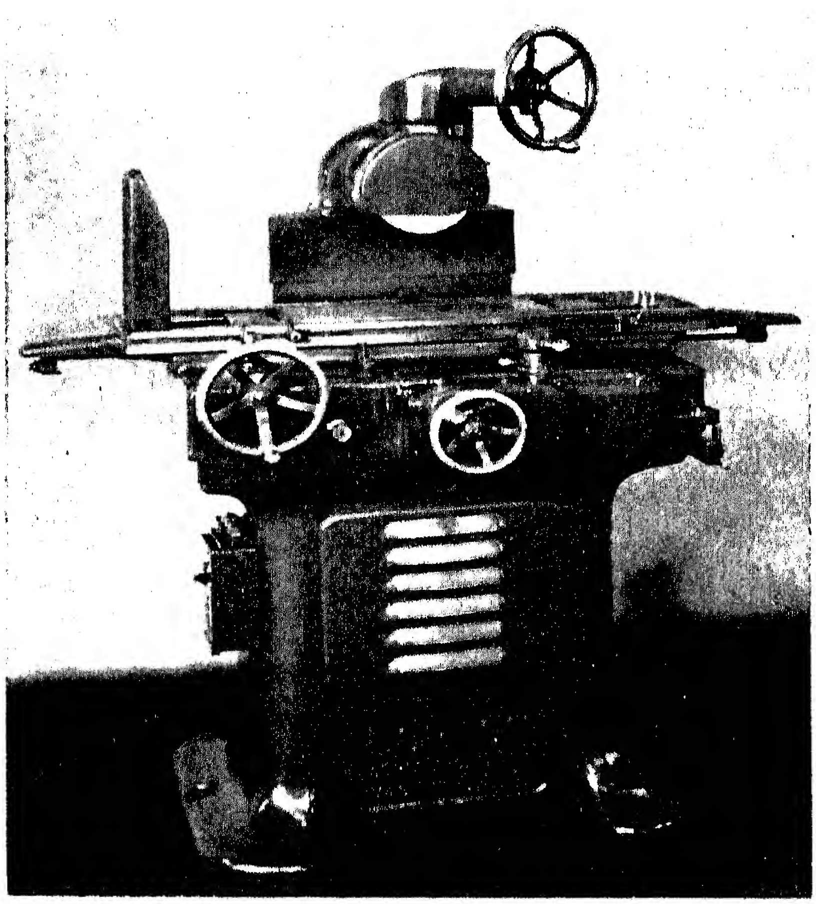
Internal planetary grinder. The internal planetary grinder is used for parts too large or unwieldy to be swung conveniently. The workpiece is clamped to a table on the machine, on which it may be reciprocated over the grinding wheel but not revolved. The grinding wheel is revolved around its own axis at high speeds and is also rotated at a slower speed in an adjustable orbit around the axis of the hole to be ground.

Surface Grinders

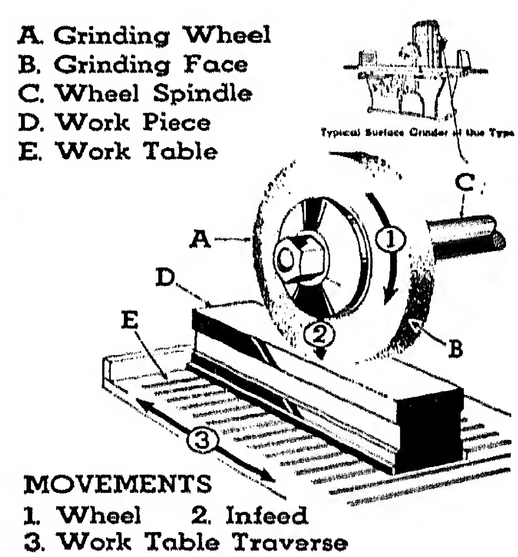
Surface grinding is concerned primarily with grinding plane or flat surfaces but also is capable of producing irregular, curved, tapered, convex, and concave surfaces.

Conventional surface grinders may be divided into two classes. One class includes planer-type machines with reciprocating tables for work ground along straight lines. This type is particularly suited to pieces that are long or have stepped or curved profiles at right angles to the direction of grinding. The second class covers the rotary-type machines with revolving worktables. Such tables can be loaded with work that is passed continuously under the wheel for rapid grinding.

Surface grinders may also be classified according to whether they have horizontal or vertical grinding wheel spindles. Grinding is done on the periphery of the wheel with a horizontal spindle. The area of contact is small and the speed uniform over the grinding



A



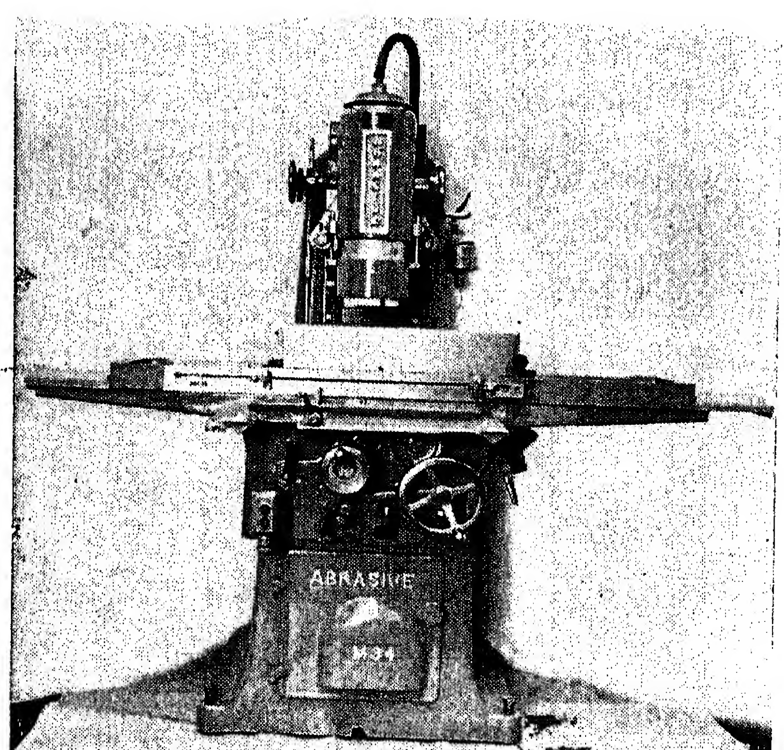
B

Fig. 17-15. Surface grinder with horizontal spindle and reciprocating table. A. A 6 in. by 18 in. surface grinder with hydraulic table traverse and automatic cross feed arranged for dry grinding. (Courtesy Norton Co.) B. Diagram of machine movements. (Courtesy The Carborundum Co.)

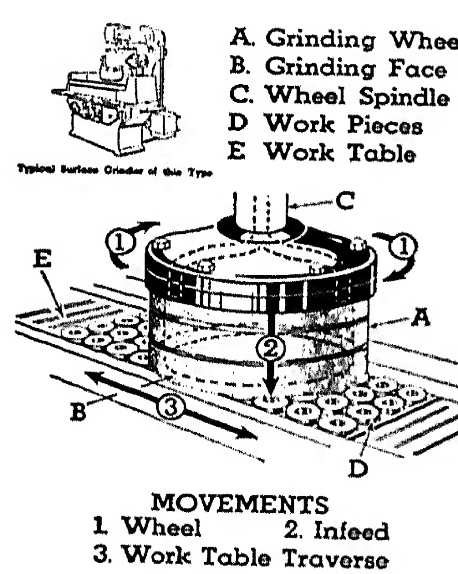
surface. Small grain wheels can be used, and the finest finishes obtained. Horizontal spindle machines are versatile and favored for doing a variety of work. Grinding is done on the side of the wheel, which may be solid, sectored, or segmental, with a vertical spindle. The area of contact may be large with face grinding and stock can be removed rapidly from the work while holding small tolerances. Relatively coarse and open grain wheels are used, and a crisscross pattern of grinding scratches is left on the work surface.

Two other types of grinders somewhat different from other surface grinders are face grinders and way grinders.

Surface grinders with reciprocating tables. A 6 by 18 in. surface grinder with a reciprocating table and a horizontal spindle is shown in Fig. 17-15 with a diagram of its movements. The machine size designates the dimensions of the working area of the table. Machines of this type for heavier work are made with tables up to 36 in. wide and 192 in. long. Grinders like the one illustrated are popular for small toolroom work. A magnetic chuck is often mounted on the table to hold work. The table is traversed longitudinally on a saddle, by hand or by a hydraulic drive reversed by dogs along the front. The saddle can be fed crosswise by hand or automatically



A



B

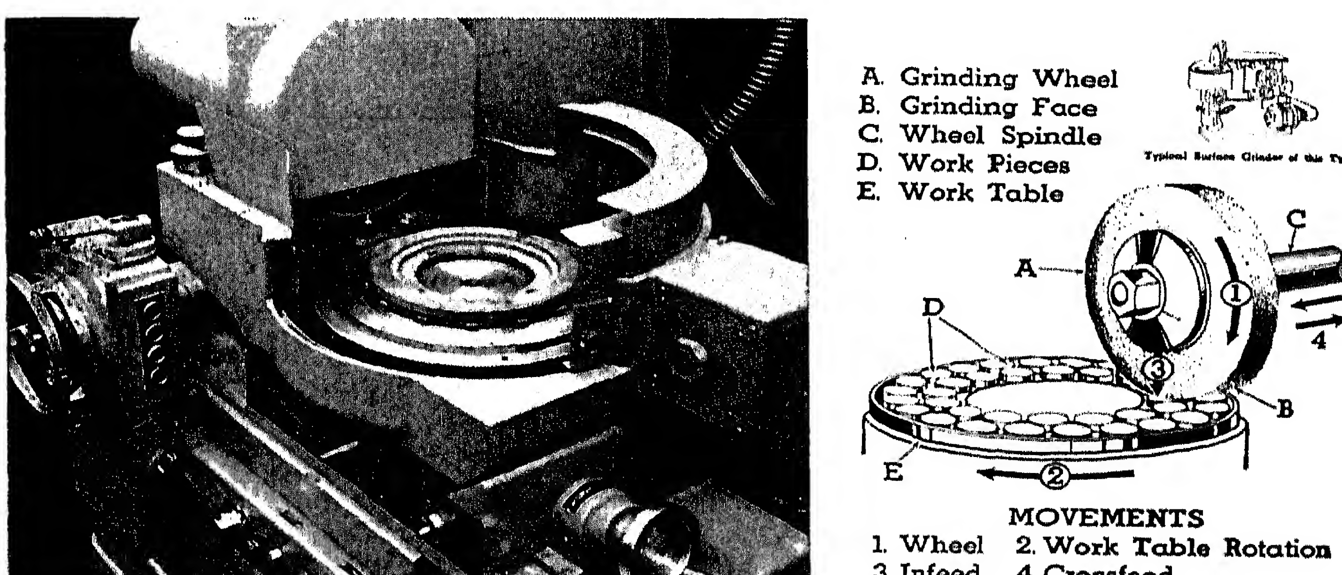
Fig. 17-16. Surface grinder with a vertical spindle and reciprocating table. A. An 8 in. by 24 in. surface grinder. (Courtesy Abrasive Machine Tool Co.) B. Diagram of machine movements. (Courtesy The Carborundum Co.)

at each table reversal, and the length of saddle feed can be limited by dogs. The wheelhead is raised or lowered on a column by means of the graduated handwheel at the top to adjust the distance between the wheel and table to grind the work to the desired thickness. The wheel is driven by a 1½ hp motor.

One form of surface grinder with a reciprocating table and horizontal spindle has its wheel cutting on the side rather than the periphery. Work carried on the table is ground on the side rather than the top. Machines like that are called *face grinders*.

An 8 by 24 in. surface grinder with a reciprocating table and a vertical spindle is shown in Fig. 17-16 with a diagram of its movements. The table reciprocates under the wheel, which normally covers all or a large part of the width of the work. The saddle can be moved crosswise by hand, and the wheelhead can be adjusted by hand or fed by power vertically. The wheelhead has a swivel adjustment to adjust the side of the wheel parallel to the table top. A 5 hp motor is built into the wheelhead and a ½ hp motor powers the table through a mechanical drive.

Surface grinders with rotary tables. A rotary table surface grinder with a horizontal wheel spindle and typical movemnts for this type of grinder are illustrated in Fig. 17-17. A round magnetic chuck on the table may carry a single piece concentric with its axis



A B

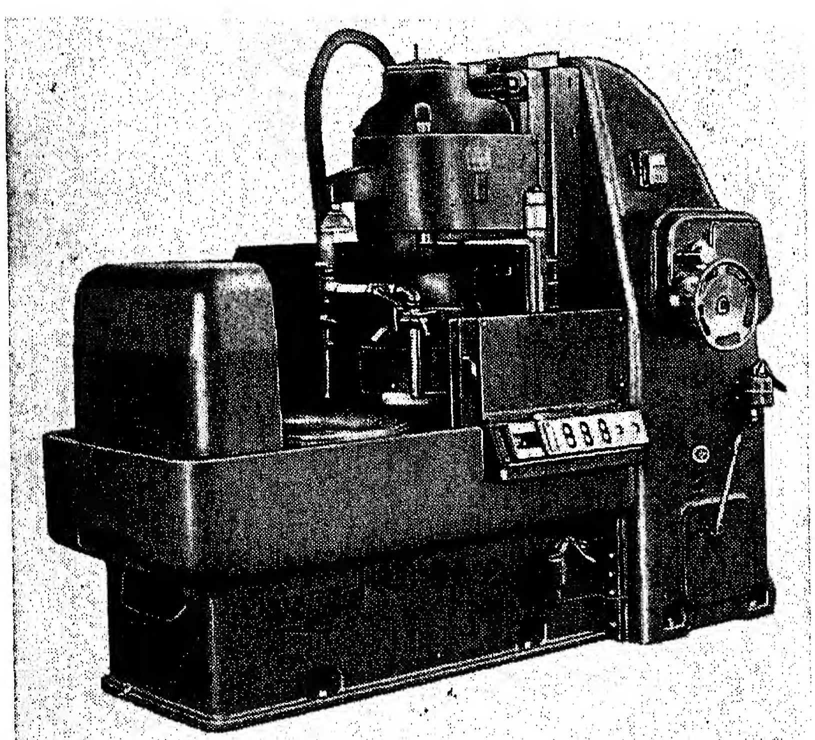
Fig. 17-17. A. A surface grinder with a horizontal spindle and a rotary table. (Courtesy The Heald Machine Co.) B. Diagram of typical machine movements. (Courtesy The Carborundum Co.)

or a number of pieces in one or more circles. Grinding marks are left on the work concentric with the axis of rotation.

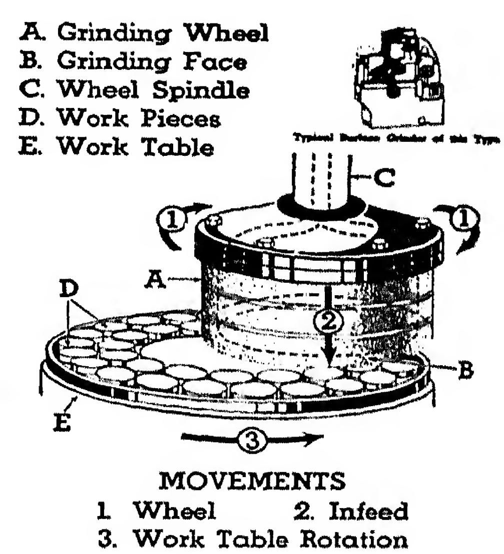
The table in Fig. 17-17 A is carried on a cradle and slide and is reciprocated under the wheel by hand or by a hydraulic drive controlled by dogs on the front of the slide. The table is tilted to grind concave, convex, or flat work. A wheel dresser is also mounted on the cradle. The wheelhead is raised or lowered on a column by means of the handwheel on the left front of the base to size the work on the table. Vertical power feed is available. Some machines of this type reciprocate the grinding wheel on a ram, like a shaper, and have a rotary table that can be raised or lowered as well as inclined.

Surface grinders of this type have been made with tables from 8 in. to 30 in. in diameter. Common sizes are 12 in. powered by a 10 hp motor and 16 in. with a 15 hp main drive motor.

The rotary surface grinder of Fig. 17-18 A has a vertical spindle driven directly by a 25 hp motor. Wheels are 16 or 18 in. in diameter. The wheelhead can be raised or lowered by hand or by power at rapid traverse or feed rates to adjust the height of the wheel over the worktable. The rotary table is a magnetic chuck and is mounted on a slide so that it can be moved to a position under the



A



B

Fig. 17-18. A. A No. 24 rotary surface grinder with vertical spindle. (Courtesy Mattison Machine Works.) B. Diagram of machine movements. (Courtesy The Carborundum Co.)

wheel, where it revolves for grinding, or away from the wheel for unloading and loading the work. Table diameters of 24, 30, and 36 in. are available.

The type of surface grinder illustrated in Fig. 17-18 is designed for production. A number of modifications and attachments have been developed for this type of machine to enhance its productiveness. Among them are automatic work loading and unloading attachments. A typical attachment feeds a steady stream of workpieces to the table rotating under the wheel where they are ground to size. A guide scrapes the finished pieces from the table and directs them down a discharge chute. A work measuring gage is often applied to indicate the size to which workpieces are being ground. A finger bears on the tops of the pieces as they emerge from under the wheel, and a dial indicator shows the heights of the surfaces above the table.

One type of rotary surface grinder with vertical spindle has two tables. Workpieces can be loaded on one table while pieces are being ground on the other table.

Disk grinders. Disk grinders remove stock rapidly by grinding with the large areas presented by the sides of disk wheels ranging in diameters from 12 in. to 72 in. The size of the machine designates the diameter of the wheel. A 3 hp motor is used to drive a 14 in. disk, and a 40 hp motor drives a 72 in. diameter disk. Exceptionally close tolerances are not generally held in disk grinding, but production rates are high. Some machines have horizontal spindles, others vertical spindles.

The single horizontal spindle disk grinder of Fig. 17-19 has a disk wheel backed by a steel plate at each end of the spindle. Work is supported on a table, adjustable for height, at each wheel. A plain table is clamped in position, and work on it is applied in a freehand manner as on the left in Fig. 17-19. A universal lever feed table oscillates on a rocker shaft to pass the work across the side of the wheel, as on the right in Fig. 17-19. The operator swings the table by means of a hand lever. Special fixtures are mounted on the slotted table top which has a movement toward the wheel regulated by a micrometer stop. The table can be tilted as much as 35° with respect to the side of the wheel.

A horizontal spindle disk grinder with two spindles has two opposed wheelheads mounted on a base. Work is fed between two

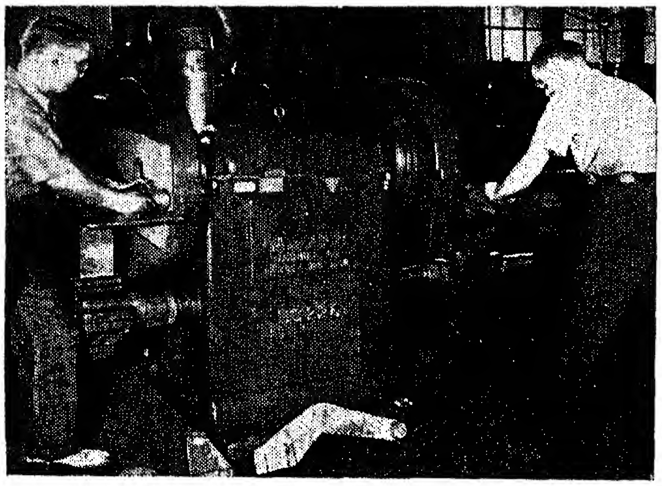


Fig. 17-19. A 26 in. single horizontal spindle disk grinder. (Courtesy Gardner Machine Co.)

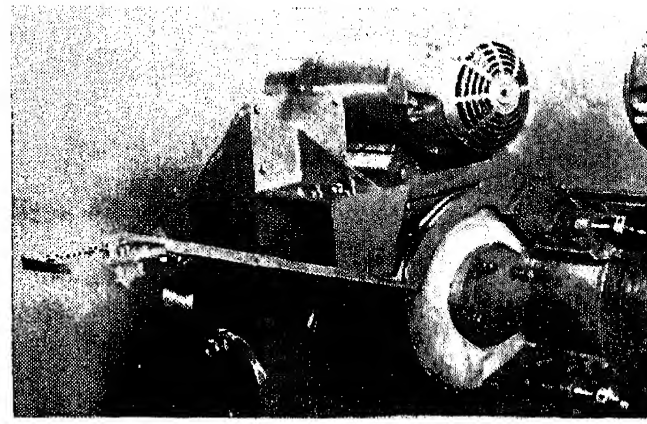


Fig. 17-20. A 15 in. double horizontal spindle disk grinder equipped with a notched rotary carrier and inclined chute to grind small sizes of ball bearing inner races at the rate of 60 to 80 per minute. (Courtesy Gardner Machine Co.)

disks, one on each wheelhead, and is ground on two sides at the same time. The distance between the disks and the parallelism of their faces are adjustable. A means is provided to feed the work between the wheels. Among the simple feeding devices are hand-operated rocking and sliding fixtures. To increase their productive capacity, these machines often have means to feed work continuously, like the rotary carrier of Fig. 17-20.

A horizontal disk grinder with a vertical spindle is shown in Fig. 17-21. Workpieces are placed on the revolving disk and stopped against the crossbar above the disk. A guard ring around the disk covers an exhaust system dust channel. Both wet and dry grinding can be done on this type of machine.

Special Purpose Grinding Machines

Thread grinders. Threads are ground for accuracy and finishes not obtainable in other ways. Tolerances have been held for size to $\pm$ 0.0001 in. per inch of pitch diameter and for lead within 0.0003 in. in 20 inches of length. Materials harder than about Rockwell C 27 can be threaded more economically by grinding than by other methods. Threads may be cut and then finish ground after heat treatment or they may be ground from solid stock.

Thread grinding is done on centertype and centerless machines.

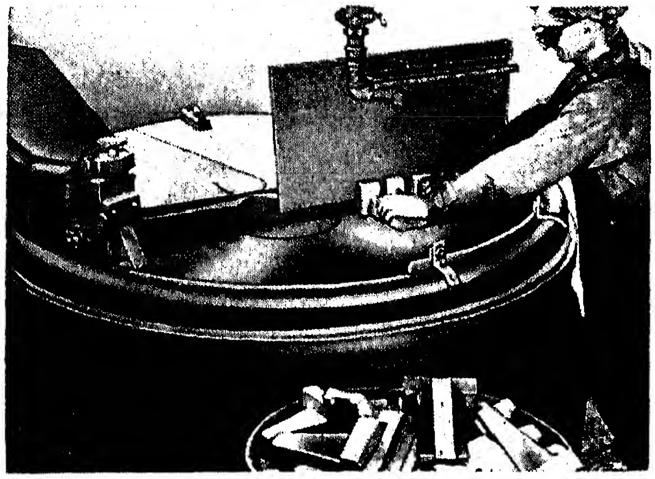


Fig. 17-21. A 53 in. horizontal disk grinder. (Courtesy Gardner Machine Co.)

Centertype machines are classed as *external, internal,* and *universal.* The universal thread grinders can grind both internal and external threads. Both thrufeed and infeed thread grinding can be done on centerless machines.

Thread grinders have single-rib-type and multi-rib-type grinding wheels. The first is a thin wheel with its outer edge trued to the shape of the thread space, the second is a wide wheel with grooves and ridges formed on its periphery.

A workpiece is mounted between centers on the swivel table of the universal thread grinder of Fig. 17-22. A master leadscrew engaged with a nut on the lower table to traverse the work is geared to the work spindle in the headstock. A variable speed hydraulic motor turns the work spindle. The relation between the work spindle rotation and table travel that determines the lead of thread ground can be changed by means of pick-off gears. The grinding wheel and its drive are carried in a cradle that can be tilted to incline the wheel to the helix angle of the thread ground, up to 45° left hand and 30° right hand.

The thread grinder of Fig 17-22 has a lead pick-up device to

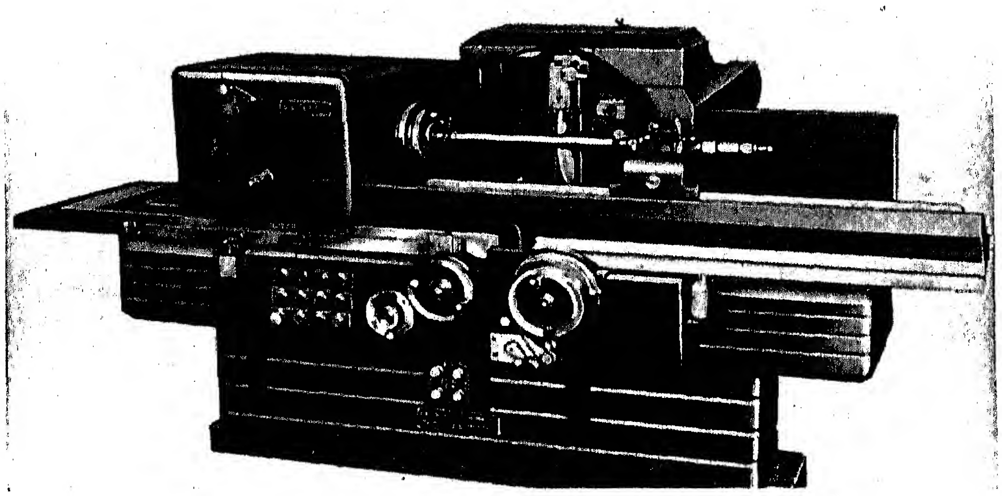


Fig. 17-22. A universal thread grinding machine for diameters up to 12 inches. (Courtesy Ex-Cell-O Corp.)

align previously roughed threads with the grinding wheel. A plain wheel dresser is available for straight line thread forms, and a universal dresser for other thread forms. The machine has an automatic cycle that can be set to take a predetermined number of cuts at specific depths and to dress the wheel before or after finish grinding.

For internal thread grinding, an auxiliary head is bolted to the front of the external spindle head, and the work is held in a chuck on the work spindle.

A centerless thread grinder resembles a conventional centerless cylindrical grinder but has provision for truing circular grooves in the grinding wheel and an inclinable work rest to align the threads on the work with the grooves on the wheel. Threads can be ground at rapid rates on the centerless thread grinder. As an example, ⅜ – 16 UNC headless screws are ground from solid hardened blanks at the rate of 60 to 70 screws per minute.

Cam and camshaft grinders. Cam grinders are used to grind various cams, camshafts, and pistons. They are essentially cylindrical centertype grinders with the headstock and footstock spindles arranged on a cradle to rock to and from the grinding wheel, as in Fig. 17-23. A master cam with a form corresponding to that required on the work is carried on the work spindle assembly and rotates in unison with the work. The master cam is held in contact with a roller and imparts the desired swinging motion to the work-

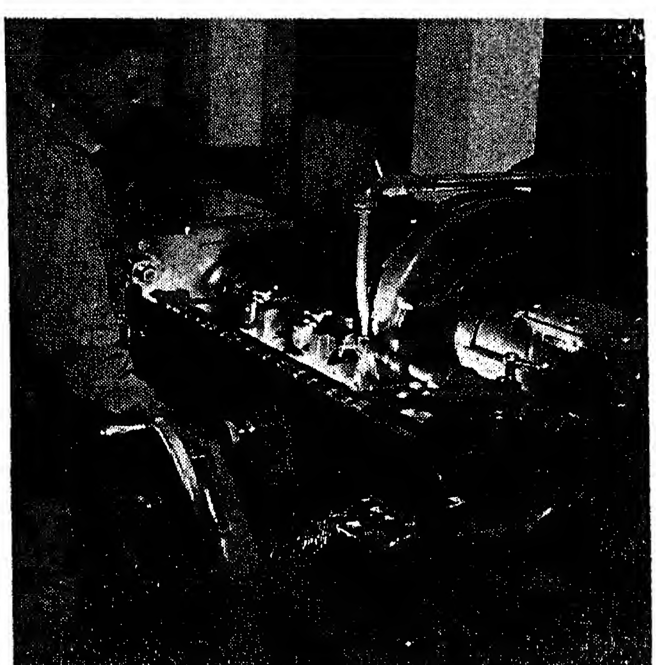


Fig. 17-23. A cam grinding machine. (Courtesy Norton Co.)

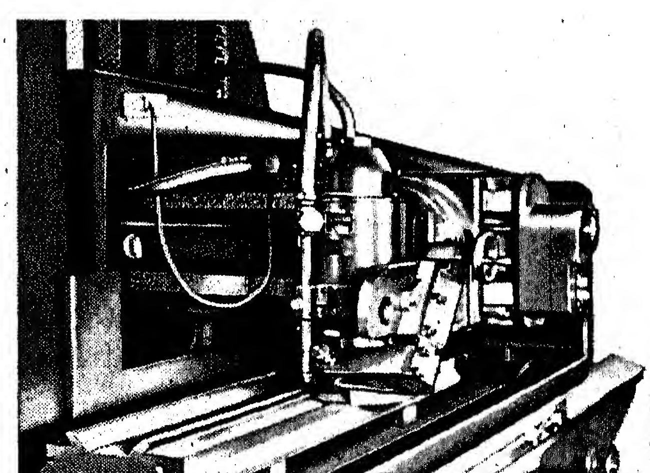


Fig. 17-24. A vertical spindle way grinder grinding the dovetail way of a milling machine table. (Courtesy Mattison Machine Works.)

piece. Cam grinding attachments that operate on the same principle are available for occasional use on cylindrical grinders where the number of cams to be ground does not warrant a special cam grinding machine.

Crankshaft grinders. Crankshaft or *crankpin grinders* resemble cylindrical centertype grinders but are implemented to grind the offset pins in the throws of crankshafts. A crankshaft is held in two indexing potchucks, one at each end, so that it can be rotated about the axes of its offset pins. The two workheads must align the ends of the crankshaft accurately and turn in unison. Interlocks are provided to prevent traverse of the table when grinding, work rotation during wheel dressing, rotation of the workheads with the work unclamped, and traverse of the table while workrest shoes are in place.

Some crankshaft grinders have a single grinding wheel and grind one pin at a time, others have two wheels to grind pins in line. One manufacturer offers sizes to swing 10, 16, and 25 inches.

Way grinders. Many machine tools have hardened ways that must be ground. Soft cast iron ways are often ground to eliminate costly hand scraping. Many way surfaces are inclined, dovetailed, or inverted and not readily reached on conventional surface grinders. Way grinders like the one in Fig. 17-24 are built specifically to grind the various kinds of way surfaces. The work is carried on a hydraulically driven reciprocating table. The wheelhead can be swiveled in a vertical plane and moved crosswise on an arm that is adjustable up or down on a column. An attachment to true the grinding wheel to the desired angle is mounted on the wheelhead. Way grinders of this kind are made to handle pieces 12 to 36 in. wide, 16 to 24 in. high, and 48 to 192 in. long.

Tool and cutter grinders. Tool and cutter grinders are used mainly to sharpen cylindrical and tapered multiple tooth cutters such as reamers, milling cutters, taps, and hobs. They can also do light surface, cylindrical, and internal grinding to finish such items as jig, fixture, die, and gage details, and mandrels, and to grind single point and formed cutting tools. The typical universal tool and cutter grinder of Fig. 17-25 has a headstock and tailstock on an upper table that swivels on a lower table and can be adjusted for straight or tapered work. The headstock can be swiveled or tilted. The lower table slides lengthwise on a saddle that can be adjusted

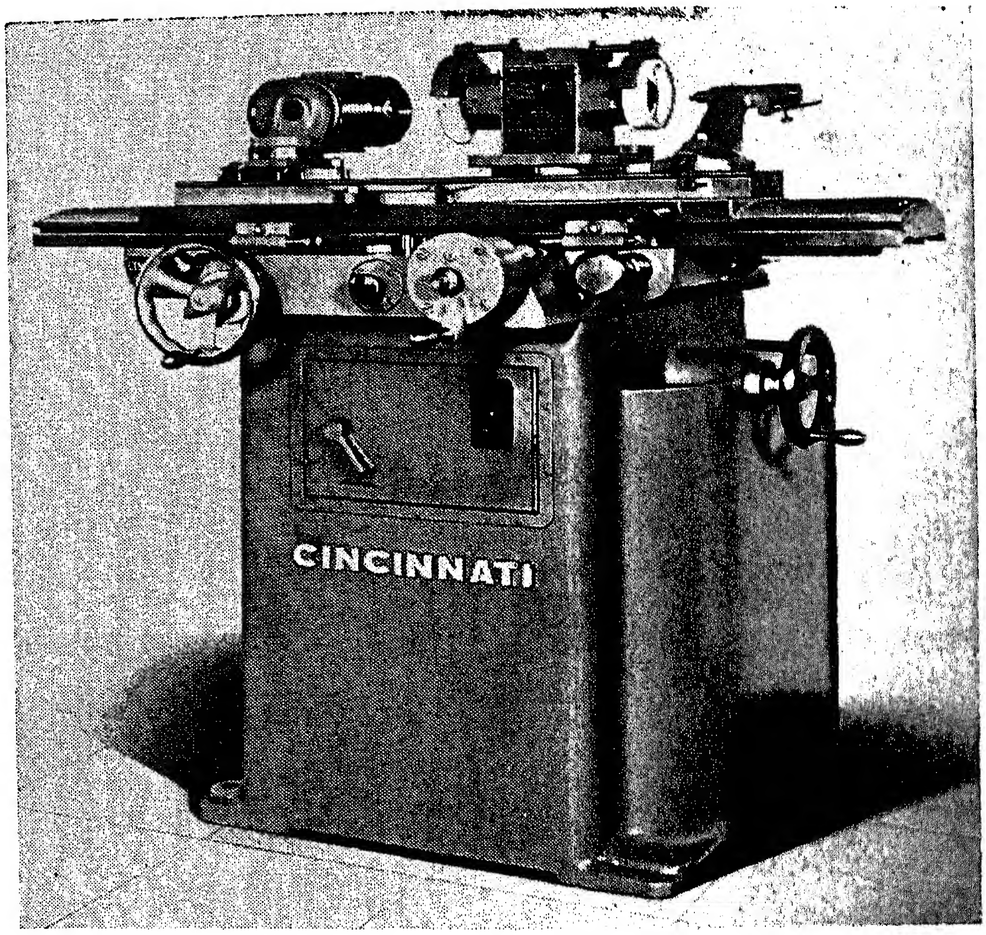


Fig. 17-25. A No. 2 tool and cutter grinder. (Courtesy The Cincinnati Milling Machine Co.)

crosswise on the bed through a micrometer dial and screw. Dogs on the front of the sliding table can be set to limit its movement. All movements except the rotation of the wheels are manual. The wheelhead carries two grinding wheels, one on each end of the spindle, and is set on top of a column so that it can be raised, lowered, or swiveled through 360°. Thus either grinding wheel can

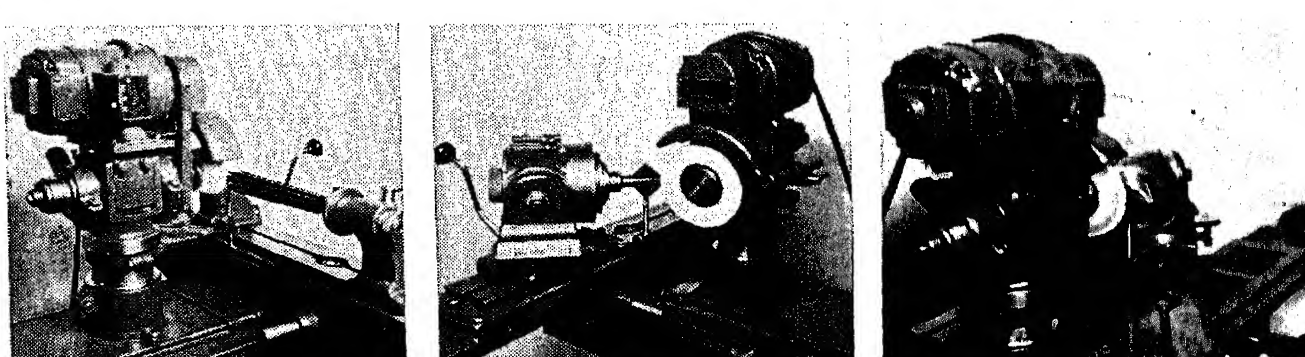


Fig. 17-26. Set ups for grinding cutters on tool and cutter grinder; a tapered reamer at left and angle cutter in middle and at right.

be set at an angle and raised or lowered with respect to the work. Several typical settings are depicted in Fig. 17-26.

The setup of a tool and cutter grinder to grind tooth surfaces at the proper angles to sharpen a multiple tooth cutter is a problem in geometry. This problem is solved by selecting a wheel of the proper shape, adjusting the various units of the machine, and arranging a suitable relationship between the cutter and grinding wheel. Many arrangements are possible on the machine to suit many kinds of cutters. A few are suggested in Fig. 17-26. Grinding is done on one tooth at a time. While being ground, each tooth is held against a thin blade on an arm called a tooth rest. As a rule, the tooth rest is mounted on the wheelhead to support and position tapered or helical teeth and on the table for straight teeth.

Other tool grinders are available, generally intended for certain classes of work. *Profile* or *contour* grinders are capable of reproducing a template form on a flat or round cutter. A type of tool grinder adaptable to a large variety of tools, but particularly to spiral tools as indicated in Fig. 17-27, is the *Monoset cutter and tool grinder.* The workhead spindle can be indexed and also synchronized with the table movement to produce helices. The workhead has a power drive, can be tilted or swiveled, and can be offset either side of center with respect to the wheelhead. The wheelhead can be moved in two horizontal directions and vertically and carries external and

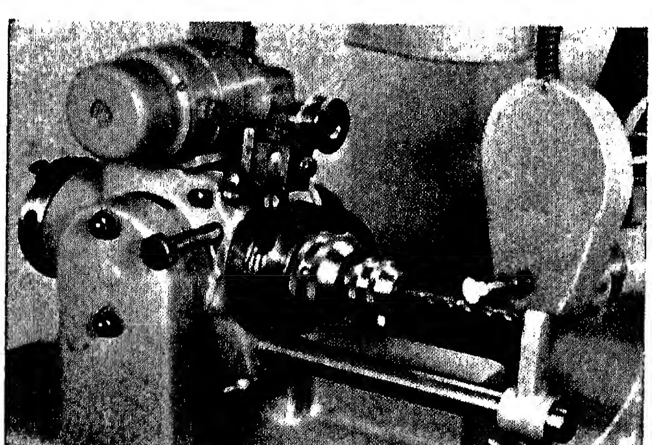


Fig. 17-27. The flutes of a twist drill being ground from the solid on a Monoset cutter and tool grinder. (Courtesy The Cincinnati Milling Machine Co.)

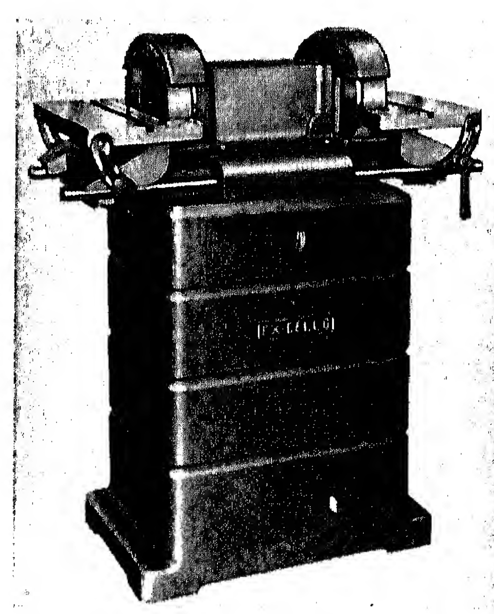


Fig. 17-28. A medium size carbide tool grinder. (Courtesy Ex-Cell-O Corp.)

internal grinding spindles. The machine enables much work to be done in one setting that otherwise would require several setups.

Carbide tool grinders. Carbide tool grinders are specifically designed for grinding the faces and radii of single point tools, especially carbide tipped tools. A typical medium-size carbide tool grinder is shown in Fig. 17-28. The tables at each end across the faces of the wheels can be tilted. The angle of inclination is determined by a graduated cam on some machines. A tool is applied to the wheel by hand while it is held on the table and guided by a protractor aligned by the keyway in the table.

Rough or Nonprecision Grinders

Snagging. Snagging operations remove sprues, gates, risers, fins, and other excess material from castings, forgings, billets, welded structures, etc. Finish is normally secondary to the removal of the metal. The ground surfaces generally need not be accurate, and the work is not located but instead frequently is presented to the

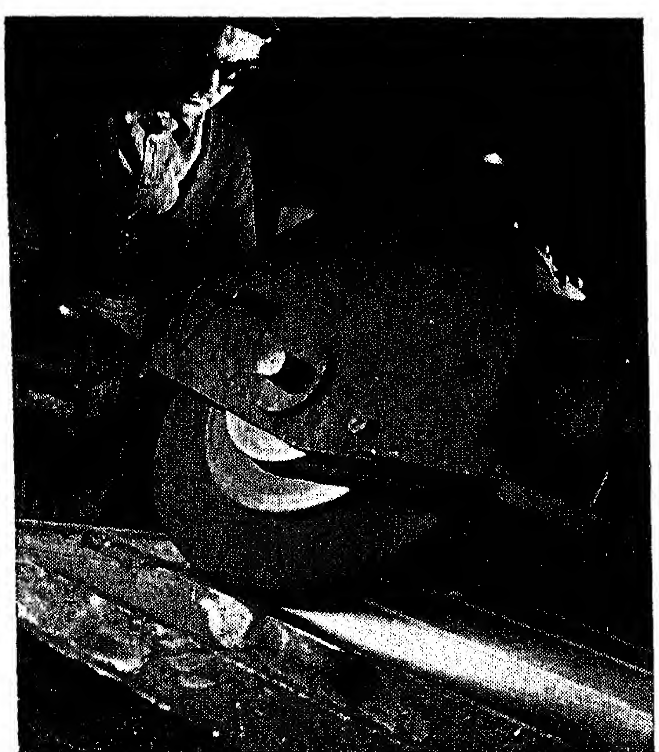


Fig. 17-29. A swing frame grinder in operation. (Courtesy The Carborundum Co.)

Fig. 17-30. Snagging a casting on a floor stand grinder. (Courtesy The Carborundum Co.)

wheel, or the wheel to the work, in an off hand manner. Machines used for snagging are swing frame, floor stand, bench, portable, and flexible shaft grinders.

Swing frame grinders. A swing frame grinder has a horizontal frame from 6 to 10 ft long suspended at its center of gravity so as to be moved freely within an area of operation. A motor on one end of the frame drives a grinding wheel on the other end through a belt. The operator applies the wheel to the work in the manner illustrated in Fig. 17-29.

Floor-stand and bench grinders. A floor-stand grinder has a horizontal spindle with wheels usually at both ends mounted on a base or pedestal. Generally the spindle is also the armature shaft of the driving motor. Work is applied to the wheels in the manner shown in Fig. 17-30. A similar form of grinder of smaller size and mounted on a bench is called a bench grinder.

Floor-stand and bench grinders are used for snagging and off hand grinding of cutting tools and miscellaneous parts. Polishing wheels may be run on these grinders for polishing and off hand surface finishing of metals.

Portable and flexible shaft grinders. The usual form of portable grinder resembles a portable drill with a guard and grinding wheel mounted on the spindle. A similar purpose machine is the flexible shaft grinder that has the grinding wheel on the end of a long flexible shaft driven by a motor on a relatively stationary stand.

Heavy portable and flexible shaft grinders are used largely for snagging, but small high speed models with little wheels are widely used in toolmaking and manufacturing to remove excess metal in the forms of burrs, sharp edges, and irregularities from many parts that in other respects are precision finished.

Belt grinders. Belt grinders are grinding and polishing machines that utilize continuous belts of coated abrasives. A belt grinder for rapid sanding of small and irregularly shaped work is shown in Fig. 17-31. It also has a spindle for abrasive

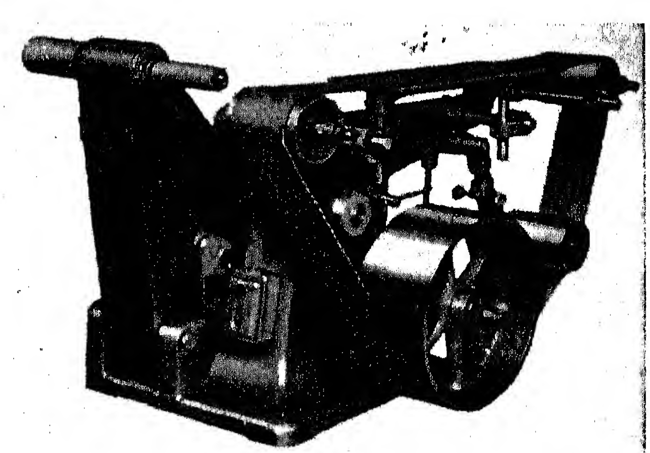


Fig. 17-31. A variety belt grinder. (Courtesy Mattison Machine Works.)

rolls. Work may be applied, commonly by hand, against the open belt, sanding pulleys, platen, shaped forms, or rolls in succession to reach various curves and flat surfaces.

Questions

1. Describe a plain cylindrical centertype grinder.
2. How does a roll grinder resemble and differ from a plain grinder?
3. Describe a universal cylindrical centertype grinder.
4. For what purposes are plain and universal cylindrical centertype grinders used? What are their relative advantages?
5. What is a chucking grinder and for what is it used?
6. Describe the principle of centerless grinding.
7. Under what conditions is a centerless grinder superior to a centertype grinder? Why?
8. Describe a typical plain internal grinder. For what is it used?
9. How may work be sized on an internal grinder?
10. Describe a centerless internal grinder.
11. What are the principal types of surface grinders? What are their relative merits?
12. What are disk grinders and how are they used?
13. What advantages do thread grinders offer?
14. Describe the action of a cam grinder.
15. Describe a typical universal tool and cutter grinder. For what is it used?
16. What is snagging? On what kinds of machines is it done?

References

Carborundum Grinding Bulletins. Niagara Falls, N.Y.: The Carborundum Co.

The Carborundum Co., Literature on operation of various types of grinding machines.

Centerless Grinder Data Book. Cincinnati, Ohio: The Cincinnati Grinders, Inc., 1929.

Heywood, Johnson, *Grinding Wheels and Their Uses.* Cleveland, Ohio: The Penton Publishing Co., 1938.

Norton Co., Worcester, Mass., Literature on operation of various types of grinding machines.

Manufacturers' Catalogs:
Abrasive Machine Co., E. Providence, R.I.

Brown and Sharpe Mfg. Co., Providence, R.I.
Bryant Chucking Grinder Co., Springfield, Vt.
Cincinnati Milling Machines and Grinders, Inc., Cincinnati, Ohio.
Ex-Cell-O Corp., Detroit, Mich.
Gardiner Machine Co., Beloit, Wis.
Heald Machine Co., Worcester, Mass.
Landis Tool Co., Waynesboro, Pa.
Mattison Machine Works, Rockford, Ill.
Norton Co., Worcester, Mass.
U.S. Electrical Tool Co., Cincinnati, Ohio.

Chapter 18

ABRASIVES, GRINDING WHEELS, AND OPERATIONS

Abrasives

ABRASIVES ARE HARD SUBSTANCES used in various forms as tools for grinding and other surface finishing operations. When applied properly, abrasives remove metal by cutting it into chips just like other metal cutting tools. The chips generally are so small that they must be magnified to be seen. Abrasives are capable of cutting materials too hard for other cutting tools. Also, abrasives remove material in relatively small amounts to attain accurate surfaces and fine finishes.

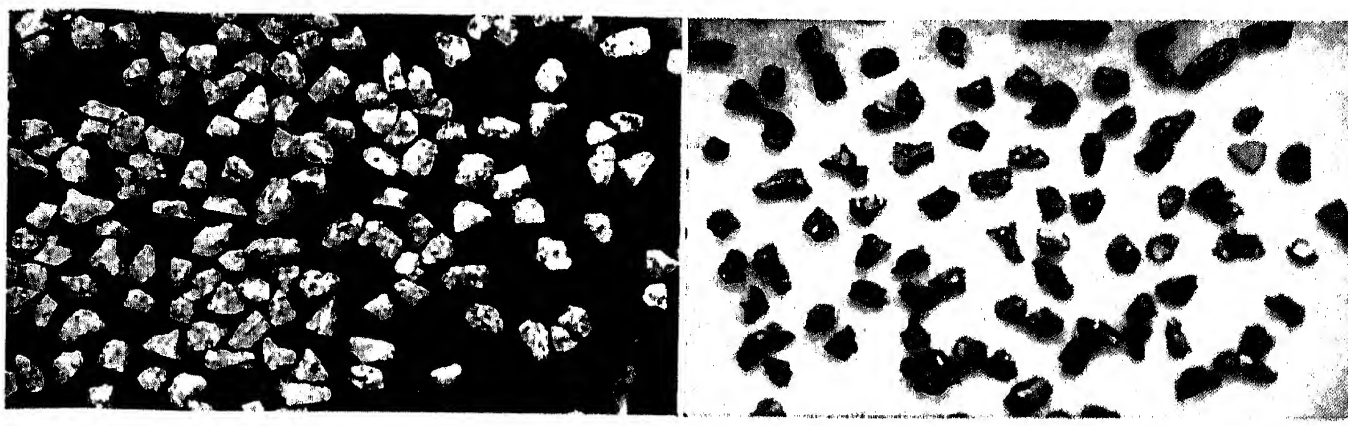
Abrasives may be used as loose grains, in grinding wheels, in stones and sticks, and as coated abrasives.

Common abrasive substances. The principal abrasive substances are:

1. aluminum oxide, chemically Al_2O_3, known by such trade names as *Alundum* and *Aloxite*
2. silicon carbide, SiC, known by such trade names as *Carborundum* and *Crystolon*
3. diamond, a form of pure carbon

Aluminum oxide and silicon carbide, illustrated in Fig. 18-1, are by far the most widely used abrasives. Variations in the composition and quality of these products are available. More than one substance is used because each has distinct properties that makes it most efficient for certain applications. The important properties of an abrasive material are (1) hardness, (2) toughness, and (3) resistance to attrition.

Hardness is the ability of a substance to resist penetration. An abrasive substance must be hard in order to penetrate and scratch the material on which it works. The greater the difference in hardness between an abrasive and the work material, the more efficient the abrasive. The diamond is the hardest known substance. If its hardness is designated by 70, then the hardness of silicon carbide may be approximately represented by 25, aluminum oxide by 20, tungsten carbide by 19, hard steel by 8, and common glass by 4. Silicon carbide and aluminum oxide are considerably harder than,



A B

Fig. 18-1. A. Aluminum oxide abrasive grains. B. Silicon carbide abrasive grains. (Courtesy Norton Co.)

and therefore suitable abrasives for, most materials.

Abrasive grains deteriorate in service by having relatively large pieces of the crystals break off and by having the sharp cutting edges become rounded. The first action is called fracture, the second attrition. The toughness or body strength of an abrasive grain is a measure of its resistance to fracture. The ability of a grain to remain sharp depends upon its resistance to attrition.

Manufacture of abrasives. Natural abrasives were all that were available until about the beginning of this century. Impure aluminum oxide occurs naturally as corundum and emery. However, natural abrasives lack uniformity and reliability and have been replaced largely by manufactured abrasives. Some natural sandstones still are used for hand-operated grindstones. Diamonds are natural.

Silicon carbide is made from sand, coke, sawdust, and salt, mixed together and piled around a carbon electrical conductor. A wall of

uncemented bricks is built around the mass, and a heavy current is passed through the electrode. A temperature around 4200° F is generated to make the silicon of the sand combine with the carbon of the coke to form SiC. The sawdust burns and leaves porous openings to let gas escape. The salt helps remove impurities. After the process has run its course, the furnace is cooled. The partially converted ingredients on the outside of the pile are removed to expose a core of loosely knit silicon carbide crystals, which is broken into individual grains.

Aluminum oxide abrasive is derived from an ore called bauxite, a claylike alumnium hydroxide. The ore is calcined to drive off excess water and then put in an arc-type electric furnace and exposed to high temperatures. Iron chips and coke are added to combine with and remove impurities. The refined aluminum oxide, ordinarily about 95 per cent pure, comes out of the furnace in a large lump called a pig. The aluminum oxide is crushed and rolled into small grains, treated magnetically to remove ferrous impurities, and washed.

Grain size. Abrasive grains must be sorted accurately into various sizes to assure that the product is uniform and dependable. The grain sizes are sorted by passing the grains through screens in mechanical sieving machines. The size of a grain is designated by the mesh of the screen through which it just passes. Thus, a grain of 20 grit passes through a mesh of 20 *openings per linear inch* but will not go through the next smaller screen. Standard screened grain sizes for aluminum oxide and silicon carbide run from 4 to 220 grits. Finer sizes, called flours, are segregated by flotation methods.

Grinding Wheels

Properties of grinding wheels. A grinding wheel is made of abrasive grains held together by a *bond.* The properties of a wheel that determine how it cuts are the kind and size of abrasive, how closely the grains are packed together, and the kind and amount of bonding material.

The principal bonds are vitrified, silicate, resinoid, rubber, and shellac bonds.

A *vitrified bond* is a clay bond fired and melted in a kiln to a glasslike consistency. It can be made strong and porous to remove stock rapidly and is not affected by water, oils, acids, or common temperature conditions. Most grinding wheels have vitrified bonds.

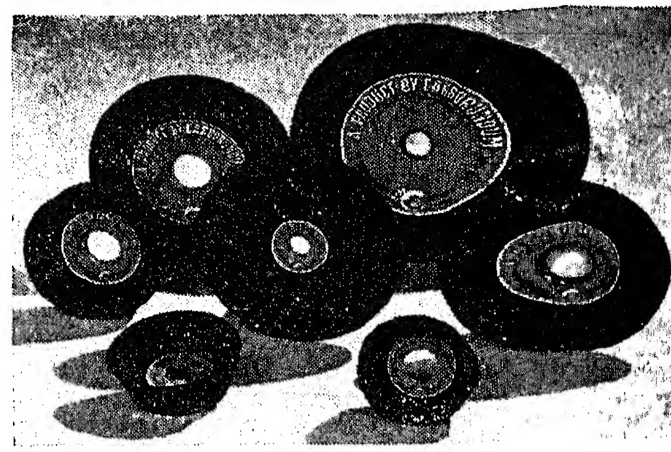


Fig. 18-2. A few typical grinding wheels. (Courtesy The Carborundum Co.)

A *silicate bond* is essentially water glass hardened by baking. It is more friable than a vitrified bond and gives a cooler cut.

A *resinoid bond* is a synthetic organic or plastic compound. It is strong and fairly flexible, can be run at high speeds, and is cool cutting.

A *rubber bond* is composed of fairly hard vulcanized rubber. Grinding wheels with rubber bond are strong and dense and can be made very thin.

A *shellac bond* helps produce high finishes on such products as cam shafts and mill rolls. It cuts coolly on hardened steel and thin sections.

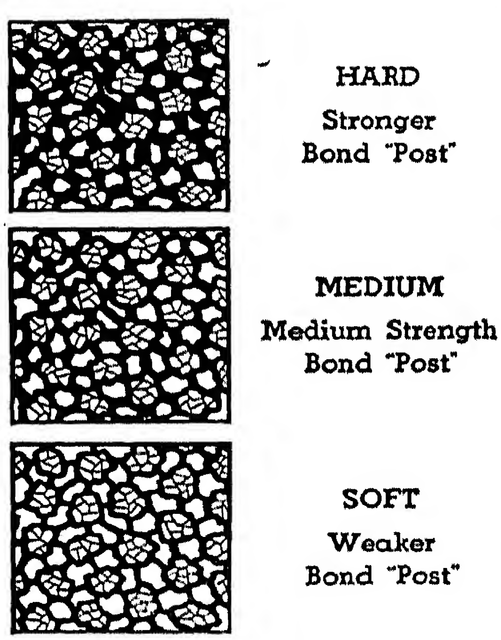


Fig. 18-3. An explanation of the meaning of wheel grade. (Courtesy The Carborundum Co.)

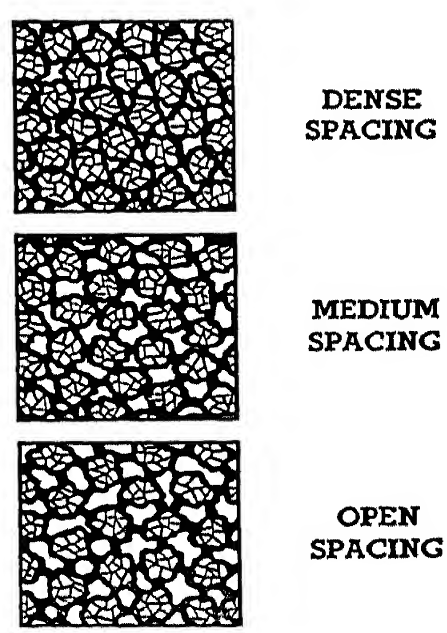


Fig. 18-4. An explanation of the meaning of wheel structure. (Courtesy The Carborundum Co.)

The *grade* of a grinding wheel is a measure of how strongly the grains are held by the bond. The bonding material in a wheel surrounds the individual grains and links them together by connectors called *posts,* as illustrated in Fig. 18-3. The sizes and strengths of the posts depend upon the kind and amount of bonding material in a wheel. The ability to hold its abrasive grains is called the *hardness* of a grinding wheel. A hard wheel holds its grains more tenaciously than a soft wheel. A wheel that is too hard for a job keeps its grains after they have become dull. A wheel that is too soft loses grains before their cutting capacities have been fully utilized. Hardness of the wheel should not be confused with hardness of the abrasive grains themselves.

The *structure* or *spacing* of a grinding wheel refers to the relationship of abrasive grains to bonding material and of those two elements to the voids between them. The meaning of structure is illustrated in Fig. 18-4. The spaces in a grinding wheel provide room for chips to escape during a cut and for cutting fluid to be carried into a cut.

Grinding wheel marking system. Grinding wheels are marked with symbols that designate their properties. A typical wheel marking and an explanation of its meaning are shown in Fig. 18-5. Usually the first symbol designates the type of abrasive, but sometimes it is preceded by a symbol to show the grain type, which specifies some marked grain characteristic such as unusual sharpness or toughness. In Fig. 18-5 the abrasive type symbol *A* for aluminum oxide is preceded by the grain type symbol *G*.

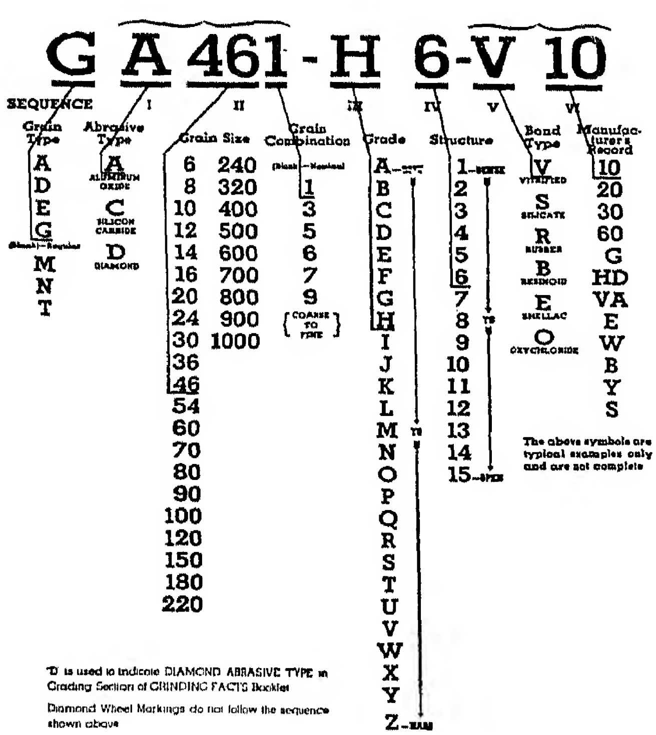


Fig. 18-5. A standard marking system chart for grinding wheels. (Courtesy The Carborundum Co.)

Grain size is indicated by a number in position II in Fig. 18-5. A 46 grit is specified as the main ingredient, but the suffix 1 designates that some grains of other sizes have been added.

Grade is designated in position III. In the specified case, the grade is *H* which stands for

medium soft. Structure is specified in position IV. In this case the number 6 denotes a close to medium spacing. The bond type is shown by a letter in position V. The example stands for a vitrified wheel. The manufacturer inserts symbols in position VI to identify his private records.

All grinding wheel manufacturers use substantially the same standard wheel marking system. However, properties of wheels are determined to a large extent by the ways the wheels are made. The processes vary from one plant to another, and wheels carrying the same symbols but made by different manufacturers are not necessarily identical.

Wheel shapes and sizes. Nine grinding wheel shapes recognized as standard are shown in Fig. 18-6. The dimensional sizes in which these wheel shapes are available also have been standardized.

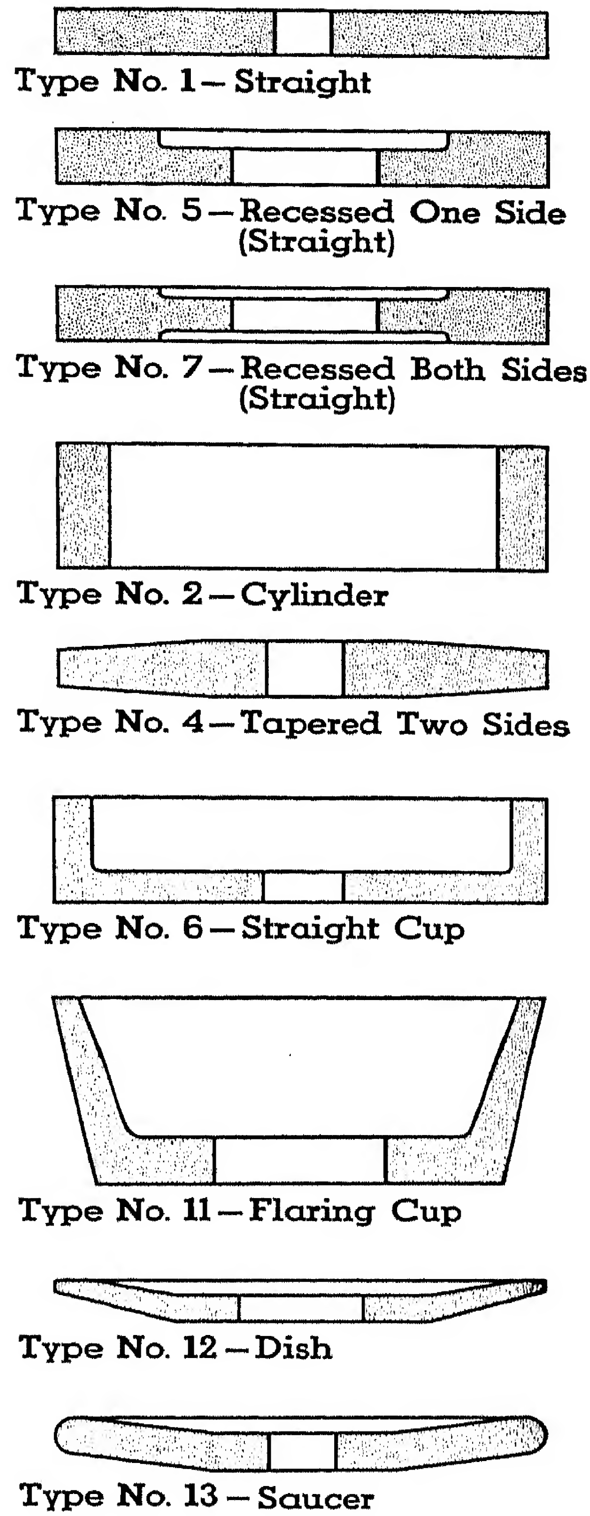


Fig. 18-6. Standard grinding wheel shapes. (Courtesy The Carborundum Co.)

Wheel type Nos. 1, 5, and 7 are commonly applied to internal, cylindrical, horizontal spindle surface, tool, and off hand grinding and snagging. The recesses in type Nos. 5 and 7 accommodate mounting flanges. Type No. 1 wheels from 0.006 to ⅛ in. thick are used for cutting off and slotting.

Wheel type No. 2 may be arranged for grinding on either the periphery or side of the wheel.

Wheel type No. 4 takes tapered safety flanges to keep pieces from flying if the wheel is broken while snagging.

The straight cup wheel type No. 6 is used primarily for surface

grinding, but also for off hand grinding of flat surfaces. Plain or beveled faces are available.

The flaring cup wheel type No. 11 is commonly used for tool grinding. With a resinoid bond, it is useful for snagging. Its face may be plain or beveled.

The chief use of the dish wheel type No. 12 is in tool work. Its thin edge can be inserted into narrow places, and it is convenient for grinding the faces of form relieved milling cutters and broaches.

The saucer wheel type No. 13 is also known as a saw gummer because it is used for sharpening saws.

Unless otherwise specified, a straight grinding wheel is furnished with the face shown in Fig. 18-7 A. Eleven other wheel face shapes

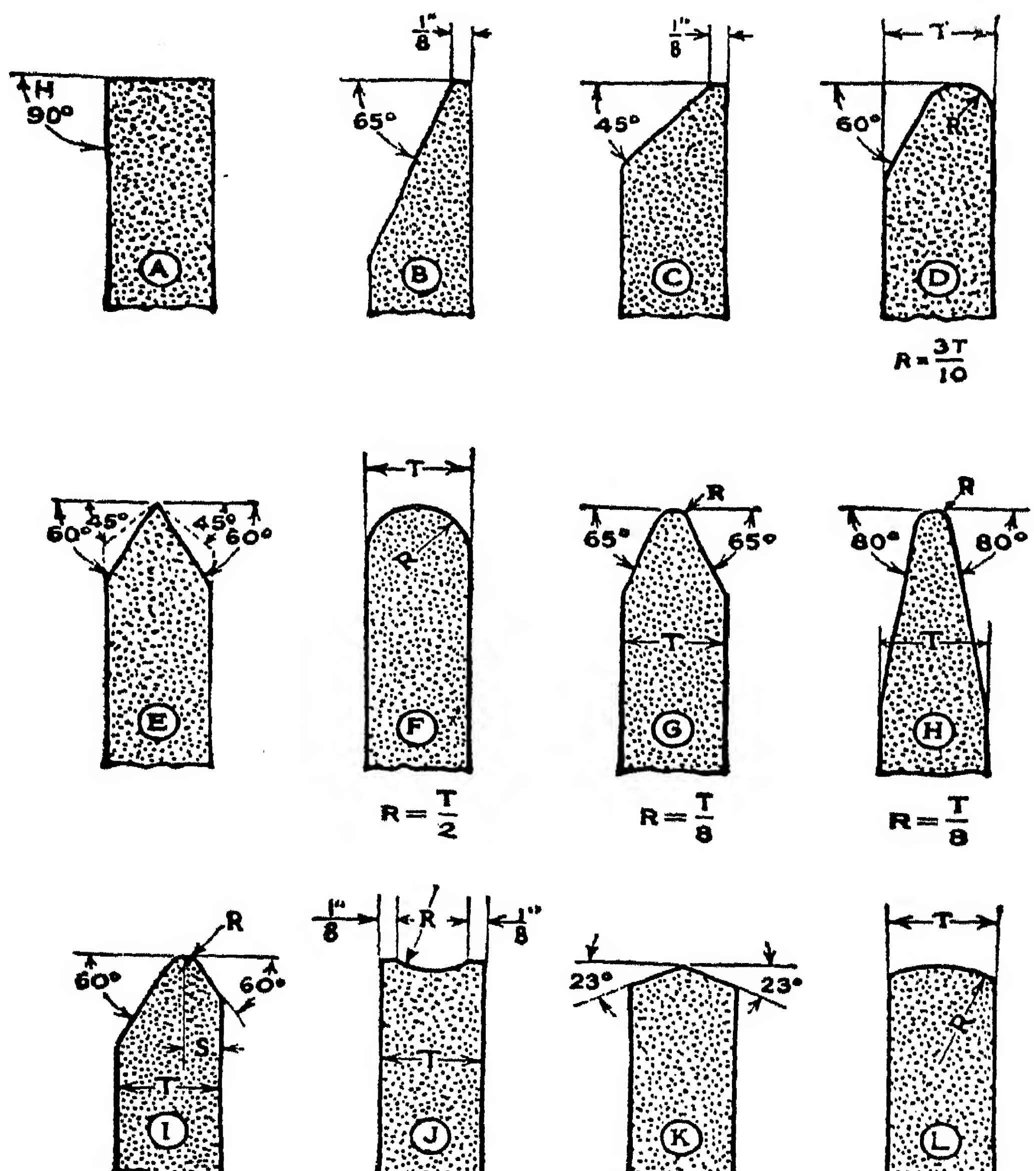


Fig. 18-7. Standard grinding wheel faces. (Courtesy The Carborundum Co.)

are available on new wheels, as shown in Fig. 18-7, and wheels may be trued to these and many other forms.

A *built-up wheel* has bonded abrasive blocks fastened to a metal wheel by means of wedges, steel bands, wire, or bolts. Built-up wheels are large in size, up to 15 in. wide by 72 in. in diameter. A segmental wheel for surface grinding with the side of the wheel consists of formed abrasive blocks held in a circle by a chuck. These wheels are easier to make than solid wheels in large sizes and have an intermittent cutting action that promotes cool grinding.

Mounted wheels and points. Mounted wheels and points are small grinding wheels with attached shanks. A few examples of the many sizes and shapes available are shown in Fig. 18-8. They are revolved at high speeds, up to 100,000 rpm depending upon size, usually on portable grinders and are used for burring, removing excess material from dies and molds, grinding in recesses and crevices, and for small holes.

Manufacture of grinding wheels. Vitrified grinding wheels may be made by the *puddled process* or *pressed process.* Pressed wheels are denser than puddled wheels. First clay and abrasive are mixed thoroughly by machine. In the puddled process, water is added, and the mixture is poured into molds. For pressed wheels, the dry or semidry mixture is placed in molds and squeezed in hydraulic presses. At this stage the wheels

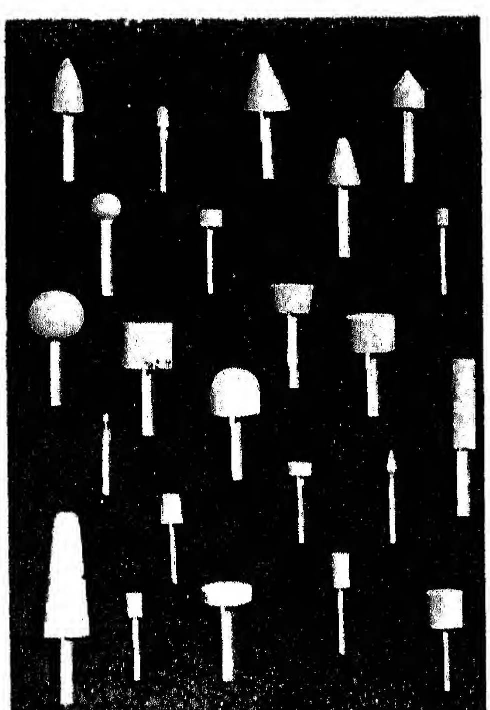


Fig. 18-8. A few examples of mounted wheels and points. (Courtesy Norton Co.)

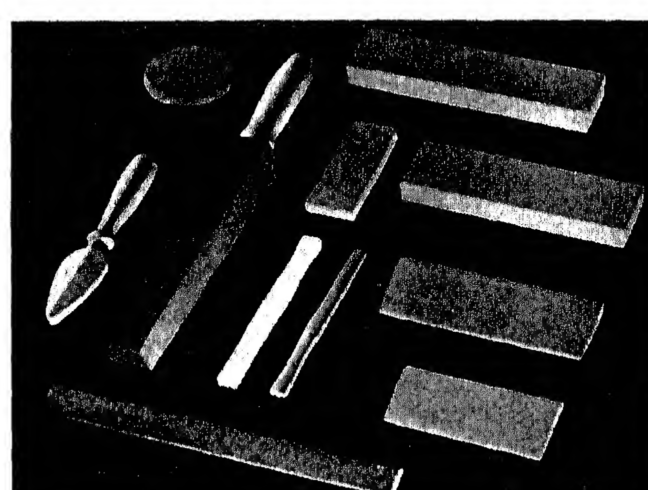


Fig. 18-9. Typical abrasive sticks and stones. (Courtesy The Carborundum Co.)

are baked and dried. The puddled wheels must be trimmed to size.

Grinding wheels are vitrified by being fired for several days at high temperatures, like pottery. When hard, the wheels are trued, their arbor holes are bushed with babbit metal or lead, and large wheels are balanced. The grade of a wheel is judged on the basis of its resistance to penetration by a tool like a screwdriver.

Abrasive and sodium silicate are mixed, tamped in a mold, and baked at about 500° F to make a silicate wheel.

The materials for shellac wheels are thoroughly stirred in a steam-heated mixer, molded, rolled or pressed to size, and baked in sand at low temperatures.

The raw rubber, sulphur vulcanizing agent, and abrasive for rubber bonded wheels are thoroughly kneaded together by passing between rolls. A calendar roll delivers the mixture in sheet form. Wheels are cut from the sheets and vulcanized by heating under pressure.

Abrasive, powdered resin, and plasticisor for resinoid wheels are mixed together, molded cold, and baked at around 300° F for from 12 hours to 4 days, depending upon wheel size.

Other Abrasive Products

Abrasive sticks and stones. Silicon carbide or aluminum oxide abrasive grains are bonded into sticks and stones of various types and sizes, like those in Fig. 18-9. They are used for honing, touching up the edges of cutting tools, and cleaning, polishing, and finishing dies, molds, and jigs.

Silicon carbide sticks and stones are fast cutting, but aluminum oxide stands up better under hard usage. Coarse grains are desirable for relatively heavy stock removal, but all coarse sticks and stones are not fast cutting because grain sharpness is as important as grain size.

Coated abrasives. Coated abrasives are made of abrasive grains, adhesive, and backing. The adhesive may be glue or resin and holds the grains together and to the backing. Paper or cloth backing is most common. The abrasive grains are placed on some grades of cloth in an electrostatic field that spaces and positions them with

their sharp edges uppermost. For general application, chiefly by hand, the cloth is made flexible after coating.

Abrasive cloth is used on all metals from soft aluminum and magnesium to hard steel. Aluminum oxide is generally preferred for metal. Abrasive cloth is available in 8 by 11 in. sheets, and in rolls, strips, belts, sleeves, cones, and disks of various sizes. A few examples appear in Fig. 18-10. Sheets and lengths from rolls and strips are applied by hand and on sanding machines. Endless belts are run on belt sanding machines. An example is given in Fig. 17-31. Cloth sleeves and cones are placed on expanding rubber mandrels or drums to polish inside surfaces and radii and to break the edges of castings. Plain disks are cemented to holders and revolved for off hand grinding and polishing. Slotted disks are revolved on the ends of shafts to break the edges of holes.

Polishing wheels. Flexible wheel bodies of cloth, leather, or wood, depending upon the work, are coated with adhesive and rolled in abrasive grains of uniform sizes. Coarse grains of about 36 to 60 grit may be selected for roughing, and fine grains up to 220 grit for finishing. After the adhesive, glue or cold cement, has dried, the abrasive layer is cracked by pounding to make it flexible. The resulting polishing wheels, like those in Fig. 18-11, are revolved at surface speeds around 7500 sfpm. After its grains have become dull and worn off, a polishing wheel is stripped and recoated.

Fig. 18-10. Some coated abrasives. (Courtesy The Carborundum Co.)

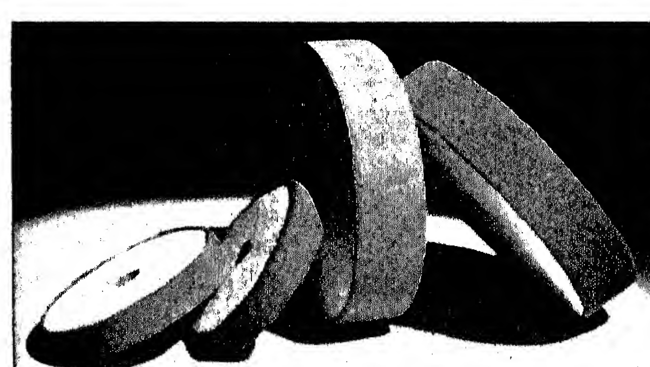


Fig. 18-11. Polishing wheels. (Courtesy Norton Co.)

Grinding Operations

Theory of grinding. Abrasive cutting edges are very small. That explains one of the advantages of grinding — that the cutting action proceeds in small stages and can be controlled to hold small tolerances and give good finishes. But appreciable quantities of material can be removed because a large number of cutting edges are applied at high frequency. For instance, about 39,000,000 cutting points act in a minute when a 46 grit wheel, 18 in. in diameter and 2 in. wide, is revolved with a surface speed of 5000 sfpm.

When abrasive grains cut, they become dull like other cutting tools. When they are dull and rounded, abrasive grains rub, create excessive heat, and smear the metal. The ideal condition is for the abrasive grains, as soon as they become dull, to break down, thereby presenting new sharp edges, or to be released from the wheel, thus uncovering fresh grains. Then the wheel remains sharp and continues to cut cleanly. However, the grains must not be cast off before they become dull because that means waste in wheel wear. Also, a wheel should wear uniformly for accurate dimensional control. Desirable grinding action produces the finish and accuracy required and removes stock as rapidly as possible.

Figure 18-12 is a simple diagram of grinding action. An abrasive grain traveling on the surface of the wheel enters the workpiece at point C, proceeds along arc CEA, and emerges at point A. The point originally at A on the workpiece has moved to B because of the rotation of the work. A chip cross section CEAB is removed. For a given wheel speed, the length of path CEA determines the amount of time the grain is in action and subject to attrition. The maximum thickness of chip BE determines the sizes of the forces that tend to fracture the grain or tear it from the wheel.

If the wheel speed is increased and the work speed is unchanged, the grain in Fig. 18-12 emerges from the work at a point between A and B. The length of the path of the grain through the workpiece is decreased but not as much as the chip thickness. Attrition of the grain is not decreased much and may be increased by the higher speed, but the forces that fracture or dislodge the grain are reduced appreciably. This and other grains are dulled more before being removed, and the wheel acts harder. On the other hand, if the wheel speed is decreased, the wheel acts softer. That may happen when a

wheel of large diameter wears to a small diameter, and then the revolutions per minute of the wheel must be increased to restore the cutting action of the wheel.

An analysis similar to that just made reveals that a decrease of work speed without change of wheel speed causes the grains to become dull before being restored and the wheel to act harder. Also, if the work speed is increased, the wheel acts softer. An increase in the depth of cut makes the wheel act softer, and a smaller depth of cut makes the wheel act harder.

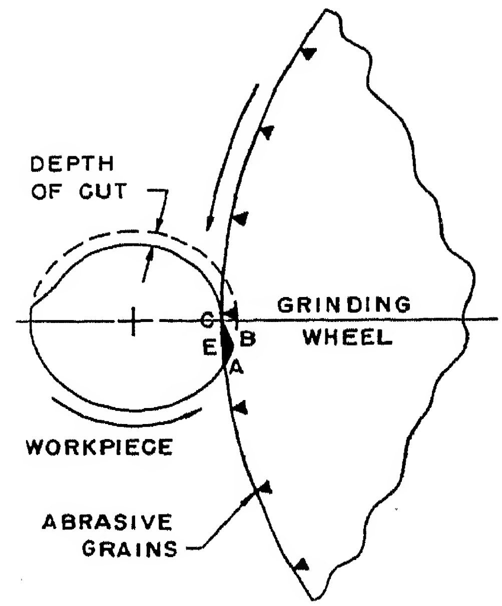


Fig. 18-12. A diagram of grinding wheel action.

The length of contact of abrasive grains with the work depends upon the type of operation. As indicated in Fig. 18-13, the length of arc is short for cylindrical grinding, longer for surface grinding with the outside of the wheel, and still longer for internal grinding. Surface grinding with the side of a wheel gives full contact. A large wheel has a longer arc of contact than a small wheel. Longer contact wears the grains faster, and the wheel acts harder. The area of contact is the product of the arc times the width of contact. As the area of contact increases, the number of grains in contact with the work increases with a uniform

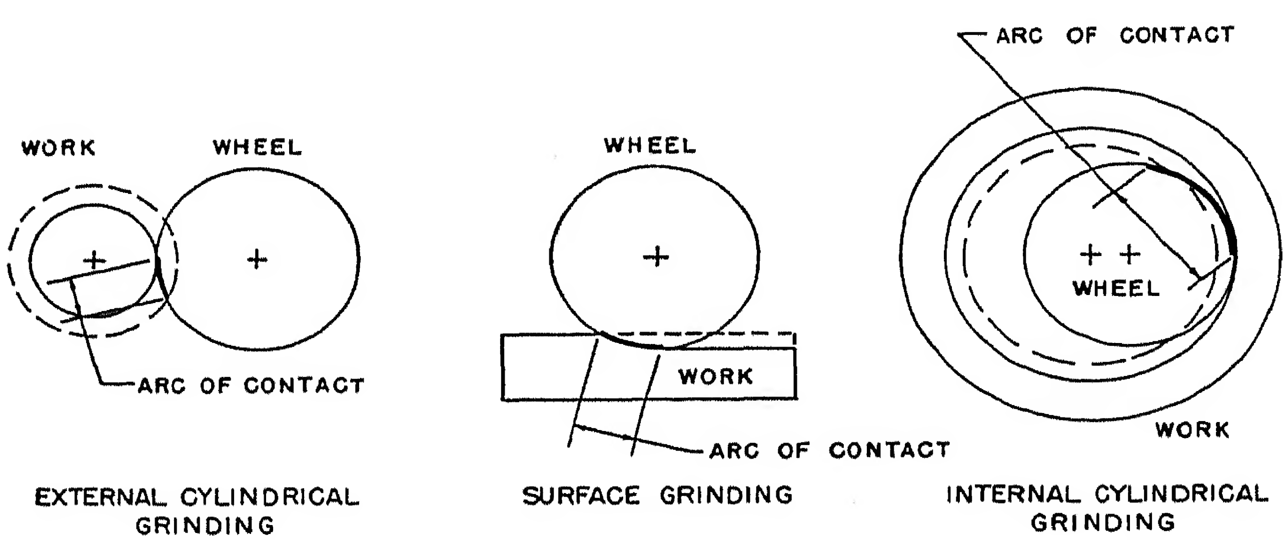


Fig. 18-13. The relation between type of operation and area of contact in grinding with the periphery of a wheel.

grain spacing, and the force tending to separate the wheel from the work is increased. That affects dimensional accuracy and finish.

The factors that must be considered in planning and executing a grinding operation are (1) the selection of the grinding machine, (2) the selection of the wheel, (3) balancing, dressing, and truing the wheel, (4) the wheel speed and work speed, (5) the rates of infeed and traverse, (6) the skill of the operator, (7) the condition of the rough workpiece, and (8) the application of cutting fluid. These factors will be discussed separately, but all are interrelated and must be harmonized for an efficient grinding operation.

The selection of the grinding machine. The principal types of grinding machines, their uses, and their relative merits are described in Chapter 17 to provide a basis for selecting machines for specific operations.

Good results cannot be expected from a grinding machine or any other machine tool in poor condition. Loose bearings and slides, misalignments, and faulty drives all contribute to dimensional inaccuracy and poor finishes on the workpiece. Vibration and looseness in a machine make a grinding wheel pound and break down rapidly.

The selection of the grinding wheel. The best results can be obtained from a grinding operation only if the proper wheel is selected. A grinding wheel may be needed to remove stock rapidly, give a high finish, or hold close tolerances. One wheel can scarcely meet all these requirements with maximum efficiency because properties that enable it to excel in one function impair it for others. A wheel should be selected for quantity production to meet the specific requirements of the job. Often rough grinding and finish grinding on a part are done in separate operations, with a different wheel for each operation. One wheel is chosen to remove stock rapidly, the other to provide the necessary finish and accuracy. For general-purpose work in small quantities, a compromise is often made by choosing a wheel that can be made to do both roughing and finishing. Grinding wheel manufacturers are represented by specialists in wheel selection and application; they should be consulted about important grinding jobs.

The properties of a grinding wheel that determine how it acts have been described. Their effects on wheel action will now be discussed.

of abrasive and grain size. Indications are that the to fracture is about the same for silicon carbide as for oxide. Silicon carbide has a low resistance to attrition ng steel, and its grains dull quickly. Aluminum oxide d for cutting steel and malleable iron. Silicon carbide etter as a rule than aluminum oxide for grinding cast , copper, soft bronze, aluminum, stone, rubber, leather, ted carbides. The reason is that silicon carbide fractures factorily in relation to the dulling of its grains, and its ges are renewed as needed. However, various kinds of oxide are made, and the crystals of some kinds are rela- k and suitable for some soft ductile materials.

finish rather than stock removal is desirable, the positions m oxide and silicon carbide may be reversed. A rapid the grains is desirable in some cases to produce fine hus, silicon carbide wheels are used to get a mirror finish ed steel rolls, and aluminum oxide grinding wheels for a on glass.

l abrasive has good body strength and a high resistance . It is far superior to other abrasive materials as far as d properties are concerned, but its cost makes it less l for most applications. Diamond abrasives are used for tting and finishing gems, ceramics, stone, and cemented

grains take large bites in soft materials and are desirable ng such stock rapidly. Only small bites can be taken in in very hard materials, for which small grains are pre- use more grains can be brought to bear on the material in ne.

ins make small scratches and generally are preferred for However, coarse grains have a free clean cutting action. ately coarse grains are desirable in some cases to hold ances and produce fine surface finishes. Surface finishes microinches rms and tolerances less than 0.0001 in. have ved on grinding jobs with fast cutting wheels of 60 to 80

in wheels can be trued to various shapes and often hold s better than coarse grain wheels. An example is found

in thread grinding, where fine grain wheels are desirable to hold the contours required in the roots of threads.

Selection of wheel bond, grade, and spacing. Most grinding wheels have a vitrified bond for strength and rigidity that help to control size and finish. Large wheels can be made more easily with a silicate bond. A silicate bond wheel has a mild cool cutting action, which is desirable for such work as grinding edged tools. Resinoid and rubber bonds make flexible wheels that can be thin for operations like cutting off. Resinoid, rubber, or shellac wheels are often best for fine finishes.

Hard tough materials dull abrasives rapidly and require soft grinding wheels that release the grains readily when they become dull. On the other hand, hard wheels are suitable for soft materials.

When the area of contact is small, few grains act and the force on each grain is relatively large. A hard wheel is desirable to hold the grains. A large area of contact calls for a soft wheel. The higher the wheel speed with respect to the work speed, the softer should be the wheel, and vice versa.

Generally, hard wheels are preferred for deep cuts and rapid stock removal. Hard wheels hold their shapes and maintain dimensional accuracy better than soft wheels. On the other hand, hard wheels promote high pressures and tend to chatter, which is not desirable for good finishes.

Heavy rigid machines vibrate less and can take softer wheels than light or run down grinders.

Open spacing of abrasive grains is desirable for heavy roughing cuts in soft materials because it permits the grain to work at their fullest possible depths and provides chip clearance between the grains. Ample chip clearance is also desirable when the contact between wheel and work is long, especially in surface grinding with the side of a cylindrical wheel or cup.

Hard materials do not allow the grains to bite deeply, and a dense spacing is desirable so that many grains can act at one time. Dense spacing is also necessary for fine finishes to place the scratches close together. A wheel with dense spacing holds its shape well and contributes to dimensional control.

Shape and size of the grinding wheel. The grinding machine determines to a large extent the size and shape of the grinding wheel. A grinding machine is normally designed to revolve a certain

diameter wheel at an efficient and safe speed. Often some dimensions of the machine, usually those of the wheel guard, limit the size of wheel that can be put on the machine. Although smaller diameters must be accepted as the wheel wears, as large a diameter as possible is desirable in a new wheel because smaller diameter wheels do not cut as efficiently, require more frequent truing, and show a larger cost per cubic inch of wheel wear. In a few cases small wheel diameters are advantageous. For example, small wheels are preferred for finishing large rolls because of the small area of contact obtainable.

The diameter of the wheel mount on the machine determines the size hole the wheel must have. For heavy wheels, the width of wheel is limited by the size and strength of spindle on the machine, and the wheel mount is designed to prevent putting on the machine wheels that are too wide and heavy.

The kind of work to be done determines the size and shape of wheel. For instance, in internal grinding the wheel must be small enough to enter the hole. Certain shapes of wheels are best for each type of grinding, such as cylindrical, surface, and tool grinding.

Balancing the grinding wheel. A grinding wheel must be in good balance to produce a good finish. Small wheels usually do not need to be balanced, but the larger the wheel and the higher the speed, the more necessary is balancing. A wheel should be mounted on the machine and trued before it is balanced. A conventional wheel mount for a straight wheel fits into the hole of the wheel and has two flanges that clamp against the sides or in the recesses of the wheel. A ring of blotting paper is placed between the flange and the side of the wheel to distribute the clamping pressure.

Most wheels are balanced statically by shifting weights on one flange of the wheel mount. The wheel is tested in various positions on a balancing stand like the one in Fig. 18-14. Running balance may be better for large wheels. A grinding wheel mount is available that automatically balances the wheel in a few seconds while it is running on the machine.

Dressing and truing the grinding wheel. A grinding wheel retains its true form and resharpens itself fully as it breaks down in an ideal grinding operation. That condition is only approached in the best actual operations, and the sharpness and form of the wheel must be restored from time to time by dressing and truing as indicated in

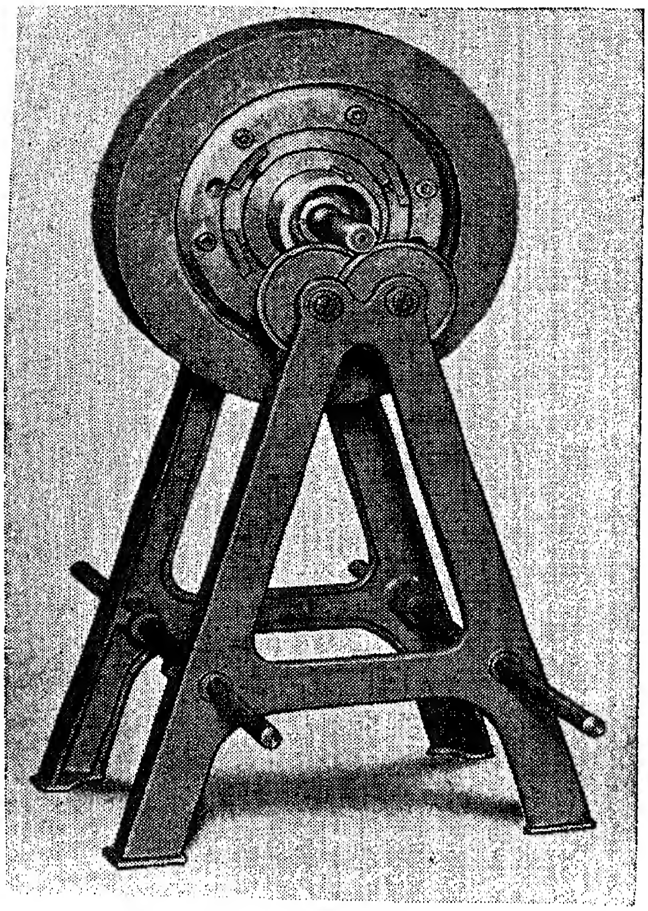


Fig. 18-14. A wheel balancing stand. (Courtesy Cincinnati Grinders, Inc.)

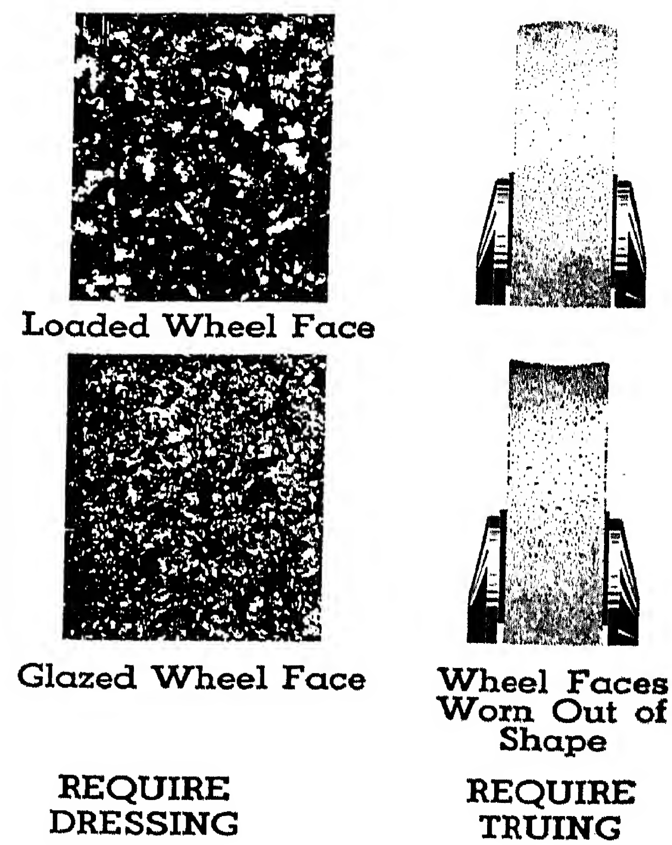
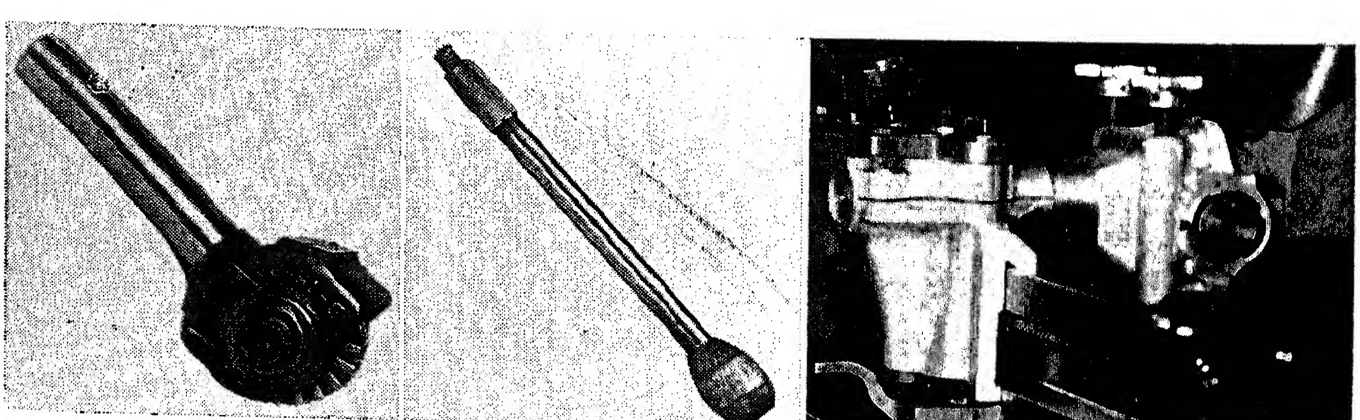


Fig. 18-15. The meaning of dressing and truing. (Courtesy The Carborundum Co.)

Fig. 18-15. How often a wheel must be dressed or trued depends upon the type of work, the fitness of the wheel, and the skill of the operator. For internal grinding, the wheel is not uncommonly trued for each piece. For some high production precision external center-



A B C

Fig. 18-16. Typical wheel dressing tools. A. A steel dressing tool. (Courtesy The Ross Mfg. Co.) B. An abrasive stick dresser. (Courtesy The Carborundum Co.) C. An abrasive wheel dresser on a cylindrical grinder. (Courtesy The Ross Mfg. Co.)

less grinding jobs, the wheels may have to be dressed only once or twice a day.

When metal particles become imbedded and fill the spaces around the surface grains, the wheel face is said to be *loaded.* Ductile materials and dense wheel structure particularly engender that condition. When the abrasive grains become dull and cease to cut efficiently because their edges are rubbed off, the wheel face becomes *glazed. Dressing* is done to restore the cutting action of the wheel by fracturing and tearing away the dull grains to expose fresh cutting edges or clear away the imbedded material. *Truing* is done to create a true surface on a wheel. The act of truing a wheel dresses it also.

Steel cutters, abrasive sticks, and small abrasive wheels are relatively inexpensive and popular tools for dressing grinding wheels. One form of steel cutter with star-shaped disks that turn freely on a shaft in a holder, like the one in Fig. 18-16 A, is efficient for removing metal from the surface of the wheel. Another form has a grooved steel cylinder in place of disks and is often used for dressing wheels for commercial grinding. Round abrasive sticks may be mounted in a holder, as in Fig. 18-16 B, and are convenient for rough profile forming and dressing thin wheels. An abrasive wheel dresser, like that in Fig. 18-16 C, puts a smooth clean-cutting face on a grinding wheel. These tools may be applied off hand or attached to holders on the machines. A wheel-type dresser turns in contact with the grinding wheel and is inclined at a small angle to give a wiping or shearing action across the face of the wheel.

Diamonds unsuited for gems are mostly used for truing wheels for precision grinding. A single large stone is set in a matrix in a holder. When worn, it is reset. Small diamonds are set in groups for efficient utilization. A diamond tool should always be applied to a grinding wheel at the angles shown in Fig. 18-17 to keep it from gouging and chattering. The stone should be rotated about its axis to a new

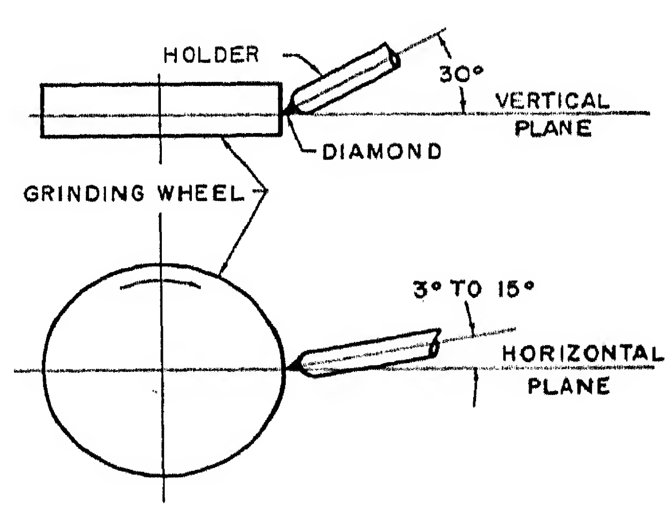


Fig. 18-17. The angles for positioning a truing diamond.

position from time to time. Light cuts of less than 0.001 in. must be taken to avoid overheating and damaging the stone. A rapid traverse across the wheel face opens the surface, a slow traverse gives a smooth surface. A coolant is used whenever possible, or else the diamond is rested often to let it cool.

Diamond tools are accurately guided and fed mechanically or hydraulically for precision truing. That can be done in most cases by means of the grinding movements of the machine. For that purpose, a standard grinder usually has a strategically placed diamond toolholder, like the one on the tailstock in Fig. 17-3, the swinging holders of Figs. 17-12 and 17-14, and the holder on the table in Fig. 17-17. Truing attachments for straight surfaces, angles, radii, and almost any other form are added to grinding machines where the normal machine movements do not suffice for truing. Truing attachments are shown on the grinders in Figs. 17-7, 17-9, and 17-24.

Crush dressing is a method of truing and dressing a grinding wheel by means of a hard roller pressed against the slowly revolving grinding wheel. The reverse of the form desired on the wheel is given to the roller, which displaces and crushes the surface grains and imprints the form on the wheel.

Crush dressing is often more economical than diamond truing, especially for intricate forms. Sharper crystals are left by crush dressing, and that means cleaner and cooler cutting. Diamond truing is generally more accurate and can be made to produce better surface finishes.

Wheel speed and work speed. The faster a wheel is run, the more efficiently it cuts, but if it is run too fast, it will fly apart. A machine may not have enough speed to revolve small or worn wheels fast enough to get the most desirable surface speeds. New grinding wheels are mostly operated at the highest attainable surface speeds that are safe. Vitrified wheels are limited to surface speeds of 6500 sfpm. Resinoid, rubber, or shellac wheels can be run at higher speeds, from 9,000 to 16,000 sfpm.

The work speed can be changed over an ample range on most grinding machines and is varied to control the grinding action. A work speed is selected to give the desired finish and highest rate of production obtainable for the workpiece material and size. For external cylindrical grinding, work speeds vary from 30 sfpm for hardened steel, which the abrasive cannot cut deeply, 50 to 100

sfpm for average work, to as high as 150 sfpm in some cases. For average internal grinding, surface speeds of 150 to 200 sfpm are advocated. Work speeds on surface grinders generally are less than 100 sfpm.

Rates of infeed and traverse. Cylindrical grinding may be done by the plunge cut or traversing methods. Similar methods are found in surface grinding. In *plunge cut grinding,* a wheel somewhat wider than the work surface is fed in as the workpiece revolves. The rate of infeed can be selected to control the grinding action, because a rapid rate makes the wheel act soft, and a slow rate makes the wheel act hard. With other factors constant, the rate of stock removal depends upon the rate of infeed. The factors that limit the rapidity of infeed and stock removal are the power of the machine, the strength of the workpiece, the structure of the wheel, and the finish desired.

A work surface wider than the wheel is *traverse ground* by traversing the work with respect to the wheel, or vice versa. Increase in the traverse rate increases the path of and the lateral forces acting on each grain, making the wheel act softer. An advance of 1/4 to 1/2 of the width of the wheel per revolution of the work in cylindrical grinding or per stroke in surface grinding is desirable to allow the rest of the wheel to clean up the surface. The leading portion of the wheel does most of the work and wears faster than the trailing portion.

If the wheel leaves the end of the workpiece in traverse grinding, the area of contact is decreased, and the grinding pressure is increased at the end of the workpiece. That causes the wheel to cut deeper and puts a taper on the end of the piece. Good practice is to allow only about a half of the wheel to travel past the end of the workpiece.

In traverse grinding, the wheel is fed in at each end of the stroke 0.001 to 0.004 in. per pass for roughing and 0.0002 to 0.002 in. per pass for finishing.

The skill of the operator. As has been described, the operator can do much to control a grinding operation. How much he takes advantage of the opportunities offered depends upon his skill. Actual tests have shown that grinding costs may differ by 100 per cent on the same job and kind of machine in the same plant with

position from time to time. Light cuts of less than 0.001 in. must be taken to avoid overheating and damaging the stone. A rapid traverse across the wheel face opens the surface, a slow traverse gives a smooth surface. A coolant is used whenever possible, or else the diamond is rested often to let it cool.

Diamond tools are accurately guided and fed mechanically or hydraulically for precision truing. That can be done in most cases by means of the grinding movements of the machine. For that purpose, a standard grinder usually has a strategically placed diamond toolholder, like the one on the tailstock in Fig. 17-3, the swinging holders of Figs. 17-12 and 17-14, and the holder on the table in Fig. 17-17. Truing attachments for straight surfaces, angles, radii, and almost any other form are added to grinding machines where the normal machine movements do not suffice for truing. Truing attachments are shown on the grinders in Figs. 17-7, 17-9, and 17-24.

Crush dressing is a method of truing and dressing a grinding wheel by means of a hard roller pressed against the slowly revolving grinding wheel. The reverse of the form desired on the wheel is given to the roller, which displaces and crushes the surface grains and imprints the form on the wheel.

Crush dressing is often more economical than diamond truing, especially for intricate forms. Sharper crystals are left by crush dressing, and that means cleaner and cooler cutting. Diamond truing is generally more accurate and can be made to produce better surface finishes.

Wheel speed and work speed. The faster a wheel is run, the more efficiently it cuts, but if it is run too fast, it will fly apart. A machine may not have enough speed to revolve small or worn wheels fast enough to get the most desirable surface speeds. New grinding wheels are mostly operated at the highest attainable surface speeds that are safe. Vitrified wheels are limited to surface speeds of 6500 sfpm. Resinoid, rubber, or shellac wheels can be run at higher speeds, from 9,000 to 16,000 sfpm.

The work speed can be changed over an ample range on most grinding machines and is varied to control the grinding action. A work speed is selected to give the desired finish and highest rate of production obtainable for the workpiece material and size. For external cylindrical grinding, work speeds vary from 30 sfpm for hardened steel, which the abrasive cannot cut deeply, 50 to 100

resembles a drill press. True center holes and centers are necessary if a workpiece is to revolve precisely.

The application of cutting fluid. Cutting fluids serve essentially the same purposes in grinding as in other machining operations. Water solutions and emulsions are widely used because of their cooling capacity. Oils are also used because of their lubricating value and ability to carry away fine particles.

Cooling is important because of the large amount of heat liberated in small areas in heavy grinding. A uniform temperature is necessary to prevent workpiece distortion, to grind to small tolerances, and to avoid excessive wheel breakdown. Central coolant systems are found in many plants because a uniform temperature is easy to maintain where a large volume of fluid is stored.

Lubricated metal chips are less inclined to stick in the spaces and load the wheel face. A particularly important function of a cutting fluid in grinding is to carry away chips and broken abrasive grains because these particles scratch and mar fine ground surfaces. Particles must not be given a chance to return with the recirculated fluid. Most grinding machines have settling tanks, but that often is not enough to remove all particles. Filters are recommended where good finishes must be produced.

The common way to apply a cutting fluid in grinding is to flood the working zone with a copious supply of fluid. To reach the actual area of contact, fluid is sometimes supplied through the pores of the wheel with the sides of the wheel coated with an impervious substance to confine the flow.

Grinding compared with other operations. Grinding is not economical for removing large amounts of material and usually is preceded by other operations that remove the bulk of the stock from rough workpieces. For example, a workpiece like the one in Fig. 8-8 is turned on the lathe from one inch diameter bar stock before the 0.753/0.752 in. diameter is ground. To remove ¼ in. of stock by grinding would be costly. Exceptions can be cited where work is ground from the solid, rather than machined in two operations, but then the stock normally is kept within 1/16 in. Such work is commonly done on disk grinders on castings with scaly hard skins.

The reason grinding is not economical for removing large amounts of material is that its power requirements, labor costs, and tool costs are high. Even with heavy cuts, about 10 hp is required to grind off

different operators. Thorough operator training is an esesntial ingredient of successful grinding.

Much is and can be done to reduce the skill required in grinding. More skill is normally required to do a variety of work with general-purpose equipment than to carry on a well-planned continuous production operation. An operator can take measures to make a wheel that is not quite right perform satisfactorily, but if a large number of pieces are to be ground and the best wheel is obtained for the job, the operator can be relieved of this responsibility. Marked advances have been made in grinders and attachments that aid in holding small tolerances in grinding. Machine controls and mechanisms are built precisely, so that when the operator makes an adjustment of, say, 0.0001 in., he can be sure of getting it

A popular aid in precision grinding is a gage with fingers that ride on a surface being ground and an indicator dial that reflects to 0.0001 in. the size being ground at any instant. That enables the operator to see what is taking place and to withdraw the wheel when the desired size is reached. Some production machines have similar gages connected to electronic controls for sizing work automatically. Devices are available for adjusting grinder tables for straight or tapered work within 0.00001 in. per inch.

The condition of the rough workpiece. The condition of the rough workpiece determines much of the success of a grinding operation. Too much stock requires an unnecessary amount of time for grinding; too little stock may mean that a piece will not clean up before finished size is reached. A case hardened piece may be left with a soft surface if too much material is removed. Warpage and runout determine to a large extent the amount of stock that must be provided for grinding. The better the control of these factors, the more efficient a grinding operation. Typical practice is to allow stock of 0.010 to 0.015 in. for rough and 0.002 to 0.005 in. on the diameter for finish cylindrical grinding, depending upon the size of the workpiece. About half as much is allowed on a side for surface grinding.

Center holes are frequently lapped to make them round, smooth, and clean, especially after heat treatment, for centertype grinding. The center hole is applied to a center shaped stone or abrasive cloth holder revolved on a drill press or on a center lapping machine that

resembles a drill press. True center holes and centers are necessary if a workpiece is to revolve precisely.

The application of cutting fluid. Cutting fluids serve essentially the same purposes in grinding as in other machining operations. Water solutions and emulsions are widely used because of their cooling capacity. Oils are also used because of their lubricating value and ability to carry away fine particles.

Cooling is important because of the large amount of heat liberated in small areas in heavy grinding. A uniform temperature is necessary to prevent workpiece distortion, to grind to small tolerances, and to avoid excessive wheel breakdown. Central coolant systems are found in many plants because a uniform temperature is easy to maintain where a large volume of fluid is stored.

Lubricated metal chips are less inclined to stick in the spaces and load the wheel face. A particularly important function of a cutting fluid in grinding is to carry away chips and broken abrasive grains because these particles scratch and mar fine ground surfaces. Particles must not be given a chance to return with the recirculated fluid. Most grinding machines have settling tanks, but that often is not enough to remove all particles. Filters are recommended where good finishes must be produced.

The common way to apply a cutting fluid in grinding is to flood the working zone with a copious supply of fluid. To reach the actual area of contact, fluid is sometimes supplied through the pores of the wheel with the sides of the wheel coated with an impervious substance to confine the flow.

Grinding compared with other operations. Grinding is not economical for removing large amounts of material and usually is preceded by other operations that remove the bulk of the stock from rough workpieces. For example, a workpiece like the one in Fig. 8-8 is turned on the lathe from one inch diameter bar stock before the 0.753/0.752 in. diameter is ground. To remove $\frac{1}{4}$ in. of stock by grinding would be costly. Exceptions can be cited where work is ground from the solid, rather than machined in two operations, but then the stock normally is kept within $\frac{1}{16}$ in. Such work is commonly done on disk grinders on castings with scaly hard skins.

The reason grinding is not economical for removing large amounts of material is that its power requirements, labor costs, and tool costs are high. Even with heavy cuts, about 10 hp is required to grind off

a cubic inch of ordinary steel per minute, whereas only about one hp is needed by a lathe tool for the same stock removal. Unless a grinder is much larger and heavier, it must take more time than a lathe, miller, broach, etc. to remove the same amount of stock. Also grinding wheels are expensive and wear faster than other types of cutting tools.

Material harder than about Rockwell C 45 can be machined efficiently only by abrasive action. Customary procedure in manufacturing hardened steel parts is to cut the material in a soft state leaving only enough stock for grinding, heat treat to the required hardness, and grind the surfaces that must be finished.

Fine finishes and accurate dimensions are more easily obtained by grinding than by nonabrasive methods. As a rule, grinding is done on both hard and soft materials when tolerances of less than 0.001 to 0.003 in. must be held. Also as indicated in Fig. 2-6, better surface finishes are produced by conventional grinding operations than by nonabrasive operations.

Tolerances as small as 0.0001 in. and finishes with readings as low as 10 microinches are regularly obtained by commercial grinding, but the cost increases as the requirements become more exacting. Some grinding is done with tolerances as small as 0.00002 in., roundness and straightness within 0.00002 in., and surface finishes better than 2 microinches, but only at the cost of special refined equipment and much skill, pains, and care. Such grinding is done on parts that cannot be treated readily by other processes because of shape or special requirements, but surfaces that must be highly refined are ordinarily finished by lapping, honing, and Superfinishing. As indicated by Fig. 2-6, these surface refinement methods normally cover ranges only exceptionally reached by grinding. Such methods are described in the next chapter.

Questions

1. What are the principal abrasives?
2. Describe the properties of an abrasive and how they affect its use?
3. How are silicon carbide and aluminum oxide made?
4. Name and describe the principal bonds for grinding wheels.
5. What is meant by the grade of a grinding wheel?
6. What is the structure of a grinding wheel?

7. Sketch and describe the use of the principal standard grinding wheel shapes.
8. Name and describe two processes for making vitrified grinding wheels.
9. What are the criteria of good grinding action?
10. Explain the effect upon the grinding action and the way a wheel acts when
    (a) wheel speed is increased
    (b) wheel speed is decreased
    (c) work speed is increased
    (d) work speed is decreased
    (e) depth of cut is increased
    (f) depth of cut is decreased
11. How is the length of contact between work and wheel related to the type of grinding operation?
12. Discuss the considerations that enter into the selection of a grinding wheel for an operation.
13. Distinguish between dressing and truing. Why are they necessary? How are they done?
14. What considerations determine wheel speed and work speed in grinding?
15. What considerations affect the rates of infeed and traverse in grinding?
16. What can be done to reduce the amount of skill required for grinding operations?
17. What determines the amount of stock needed for grinding? What amounts are commonly allowed?
18. What are the functions of a cutting fluid in grinding?
19. When is grinding uneconomical and for what purposes is it economical?

Chapter 19

SURFACE FINISHING OPERATIONS

IMPORTANT PRECISION SURFACE FINISHING OPERATIONS, in addition to grinding, are lapping, honing, and Superfinishing. Their stock removal capacity is low, but generally they are the most economical means of producing the most highly refined surfaces when preceded by preparatory operations.

When good surface appearance is desired but dimensions are not held closely, surfaces are finished by polishing, buffing, and tumbling.

Lapping

Purpose of lapping. Lapping is an abrading process that leaves fine scratches arrayed at random. Its purpose is to improve surface quality by reducing roughness, waviness, and defects, but it is not done for appearance alone because other methods are cheaper for that purpose. Lapping can produce accurate as well as smooth surfaces.

Lapping is done both by hand and by machines. Its range of usefulness is large. In some cases it may merely be an expedient to remove an occasional fault. It is a basic operation in job and tool shops and is commonly done to finish locating and wearing surfaces on precision tools and gages. Gage blocks, the standards of accuracy, are finished regularly by lapping. Machine lapping is common for production. Other typical lapping subjects are surfaces that must be liquid or gas tight without gaskets and those from which small errors must be removed, such as gear teeth.

A good finish is difficult to get on soft materials by lapping, and in metal working this operation is mostly confined to steels. If accuracy is to be maintained, stock removal must be low because it

cannot be controlled easily by lapping. As much as 0.003 in. may be taken off in roughing, to as little as 0.0001 in. in finish lapping.

How lapping is done. Loose abrasive mixed with a vehicle, bonded abrasive wheels, or coated abrasives are used for lapping. Grain sizes range from No. 120 to superfine flours depending upon the degree of finish desired. Diamond dust is effective for very hard materials. Wet lapping may be as much as six times as fast as dry lapping. Common vehicles are clear or soapy water, oils, and grease. Many commercial ready-mixed lapping compounds are available.

Loose abrasive and vehicle are spread on lapping shoes or quills, called *laps,* that are rubbed on the work. Laps are made mostly of soft close grained cast iron. The face of a lap becomes charged with imbedded abrasive particles. Grooves cut across the lap face serve to collect excess abrasive and dirt.

In lapping, the work and lap are not rigidly guided with respect to each other, and their relative movements are continually changed. In *equalizing lapping,* the work and lap mutually improve each

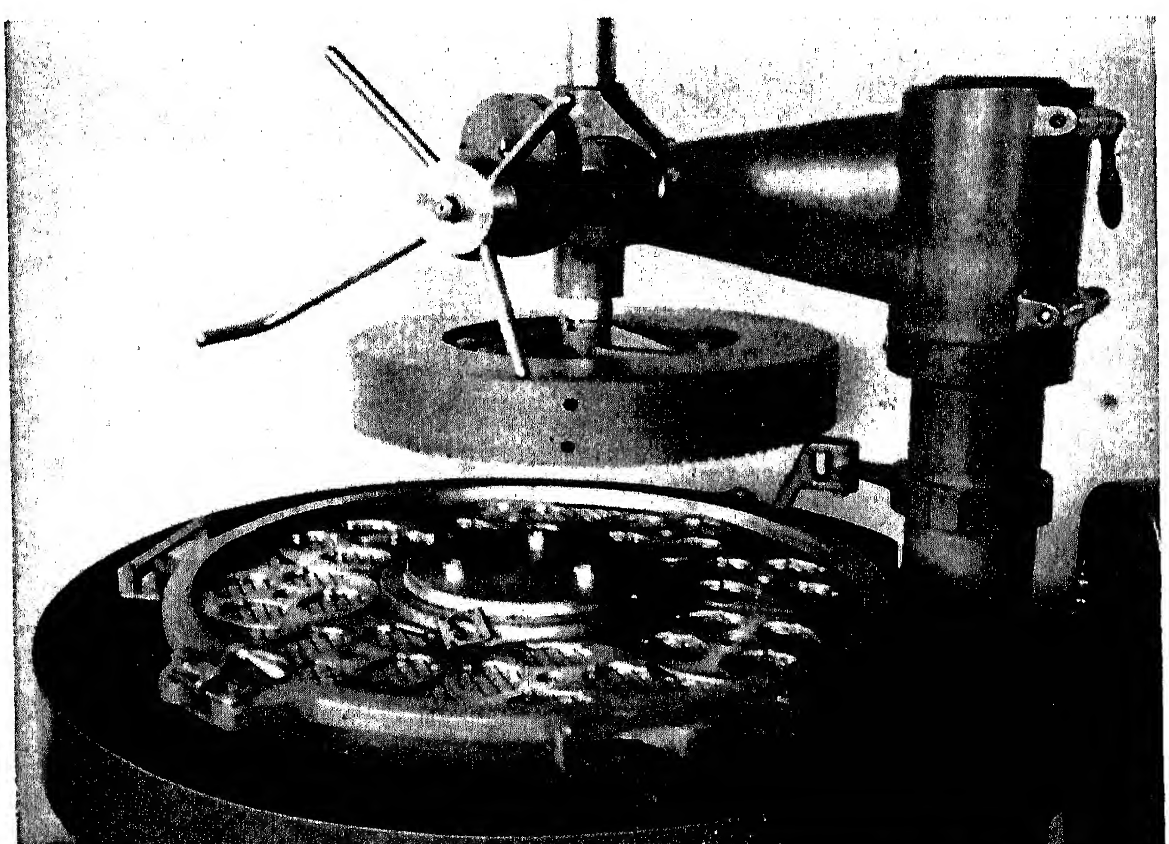


Fig. 19-1. A No. 2 vertical lapping machine with a two piece work holder and adaptors for lapping rayon pump parts. (Courtesy Norton Co.)

other's surfaces as they slide together. This is done in seating mushroom valves, machine lapping gears, and hand lapping plug or ring gages. In *forming lapping,* the work acquires a definite shape from the lap. That is the case for most lapping done with abrasive wheels.

Lapping machines. The *vertical lapping machine* of Fig. 19-1 laps flat or round surfaces betwen two opposed laps on vertical spindles. Laps from 20 to 28 in. diameter, of cast iron or bonded abrasive, are used. The lower lap revolves, and the upper one is free to float and adjust itself to the work. Speed of rotation is 45 to 65 rpm. The upper lap is suspended from an overarm and is raised to give access to the work.

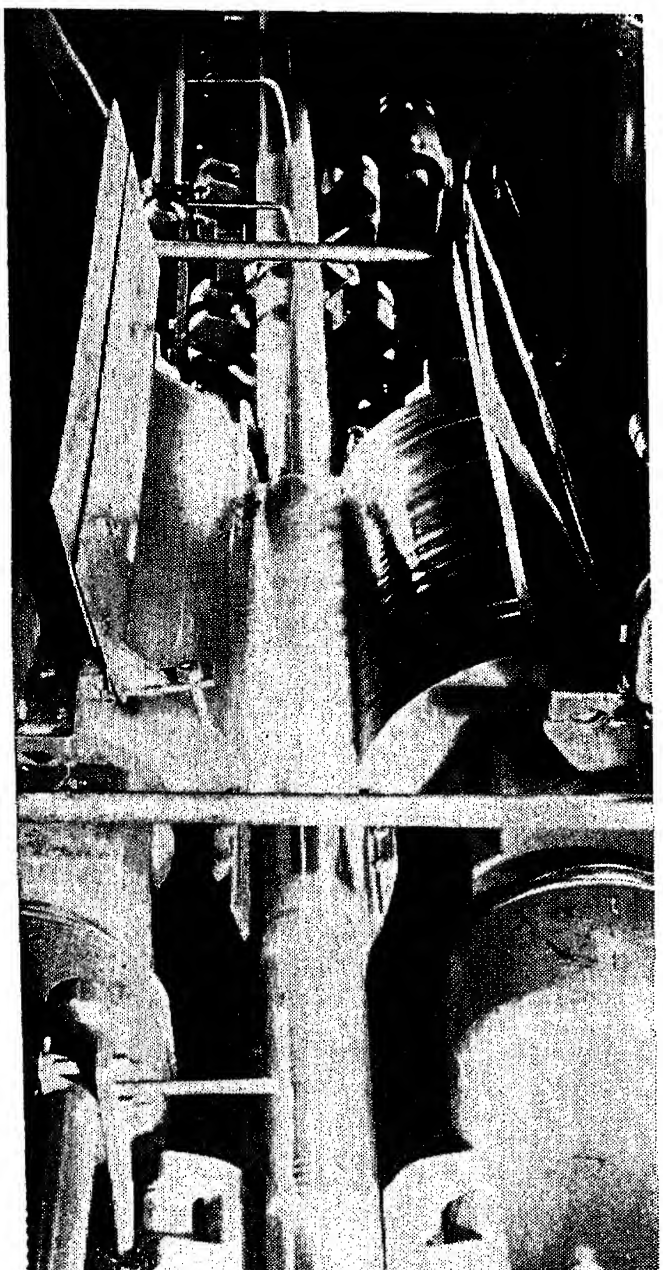


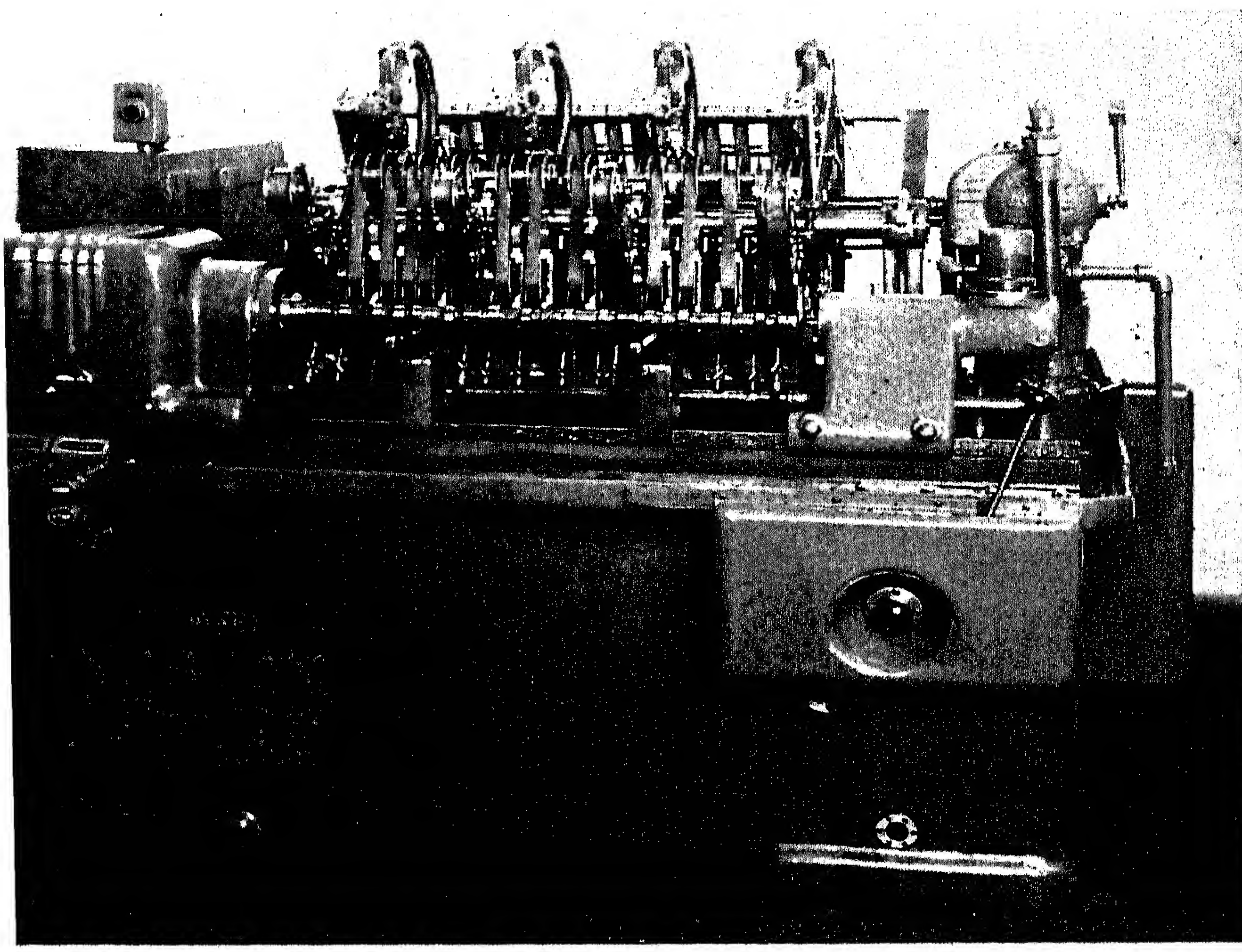
Fig. 19-2. A view of work passing between the wheels of a centerless lapping machine. (Courtesy Cincinnati Grinders, Inc.)

Cast iron laps are trued by running them together for a few moments each day. Bonded abrasive laps are diamond trued.

Workpieces are confined between the laps by a workholder and given a treatment that simulates hand lapping. Flat pieces lie in suitable openings in the workholder that is oscillated and rotated by a driving mechanism in the center. The *two-piece-type* workholder in Fig. 19-1 has circular work adapters that rotate between a hub and a split ring about the same diameter as the laps. A *plate-type* workholder is a disk of about the same diameter as the laps and has openings for flat or solid round pieces. Hollow round pieces like piston pins are carried by a *spider-type* workholder. It consists of a hub with hardened steel spindles extending at an angle from its

ery. The hollow workpieces are slipped over the spindles and d by an outer ring.

centerless lapping machine is designed for continuous produc- round parts such as piston pins, races, and cups for anti- bearings, valve tappets, and shafts. Tolerances of 0.000050 diameters and 0.000025 in. for straightness and roundness are



-3. **A No. 30 Cam-O Lap abrasive belt lapping machine for camshafts. (Courtesy Norton Co.)**

d. Surface finishes of a few microinches rms are regularly ed with a selection of lusters and finishes. The operation is ous, and as an example of output, piston pins are lapped at of 40 pins per minute.

centerless lapping machine utilizes the same principle as the ss grinder. Workpieces ride on a workrest blade as they tween two 22 in. long wheels revolving at speeds below n as depicted in Fig. 19-2. The wheels are tilted in opposite

directions to feed the work. The wheels are trued for full length contact with the work, which assures lapping action over an appreciable period of time, and wrap slightly around the pieces.

Abrasive belt lapping machines are designed to lap bearing and cam surfaces on such parts as crankshafts and camshafts that would be difficult to handle by other means. A machine of this type applies strips of abrasive coated cloth or paper to the work as shown in Fig. 19-3. The strips are wrapped partly around the surfaces treated and are held against the surfaces by shoes. A short reciprocating motion is set up between the work and strips to avoid continuous lines and refine the finish.

Honing

Purpose of honing. Honing is an abrading operation applied mostly to finishing round holes by means of bonded abrasive stones in the form of sticks. Outside surfaces are sometimes honed. Because the bonded abrasive grains do not become imbedded in the work material, soft as well as hard and nonmetallic as well as metallic surfaces can be honed. Materials honed range from plastics, silver, aluminum, brass, and cast iron to hard steel and cemented carbides. Typical applications are the finishing of automobile engine cylinders, gear bores, connecting rod bearings, gun barrels, and ring gages.

In honing, the abrasive stones are pressed against and rubbed over the work surface in an irregular path. In a hole, an expanding holder pushes the stones uniformly outward. At the beginning of the cut the stones bridge the low spots and bear upon and cut into the high spots in the work surface. As the surface is leveled off, the stones come into contact with more and more area until the entire surface receives the same abrading action.

Honing is a cutting operation and can remove stock up to 0.030 in. thick but is normally confined to amounts less than 0.010 in. To save unnecessary work, stock removal in any case should be only enough to correct and erase out-of-roundness, tool marks, crookedness, taper, and high spots plus 0.0002 to 0.001 in. on a diameter to develop the required surface finish on the base metal.

Either the honing tool or workpiece is allowed to float, their rela-

tionship is independent of the machine, and honing is thus able to correct out-of-roundness and taper but not hole location. Size is commonly controlled within 0.0001 to 0.0003 in. Surfaces can be finished to one microinch rms, but 8 to 10 microinches is the normal range. A crosshatched finish desirable for lubrication is characteristic of honed surfaces.

How honing is done. Honing stones are made from the common abrasive and bonding materials. Grain sizes range from 80 grit for roughing to 600 grit for fine finishing. Bond strength and porosity depend upon application. The stones are often impregnated with substances like sulphur, resin, or wax to improve cutting action and lengthen tool life.

Honing stones may be mounted directly or attached to metal shells or plastic tabs in holders. The stones are expanded in the work by a cone or wedges inside the holder, actuated mechanically or hydraulically. With some types of holders, an automatic size control device feeds the stones outward until the desired size is reached, allows a short dwell, and then collapses the assembly for withdrawal from the hole.

When honing is done manually, no fixture is used. The tool is rotated, and the workpiece is passed back and forth by hand over the tool. Stones are from two-thirds to twice the length of the holes.

Fairly large amounts of stock can be removed and tolerances accurately held in precision production honing on machines. The work is held in a fixture. The tool is rotated and reciprocated at the same time with the two movements purposely out of phase to cover all the surface without a regular pattern of scratches. Rotating speeds are from 15 to 20 sfpm and recipro-

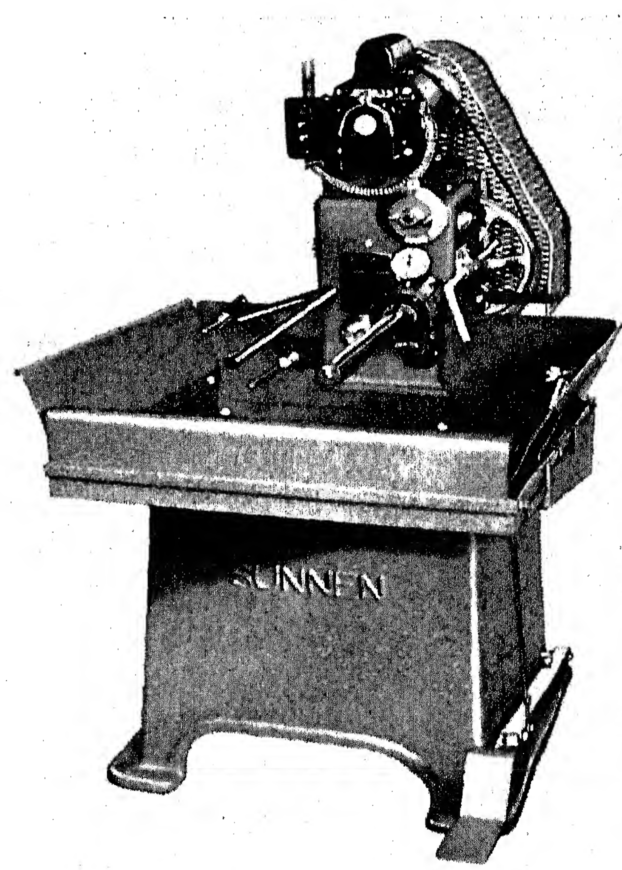


Fig. 19-4. A horizontal manually operated precision honing machine. (Courtesy Sunnen Products Co.)

cation from 20 to 90 sfpm. Too high a speed dulls the grains rapidly and causes an undesirable burnishing effect. Stones are usually about half as long as the hole and are run out the ends of the hole about one quarter to one half their length.

Honing machines. Honing is done on general-purpose machines, such as the lathe, drill press, and portable drills, as an expedient. More consistent and economical results can be obtained for production on honing machines. The two general types are the horizontal and vertical honing machines.

The *horizontal honing machine* of Fig. 19-4 has a horizontal

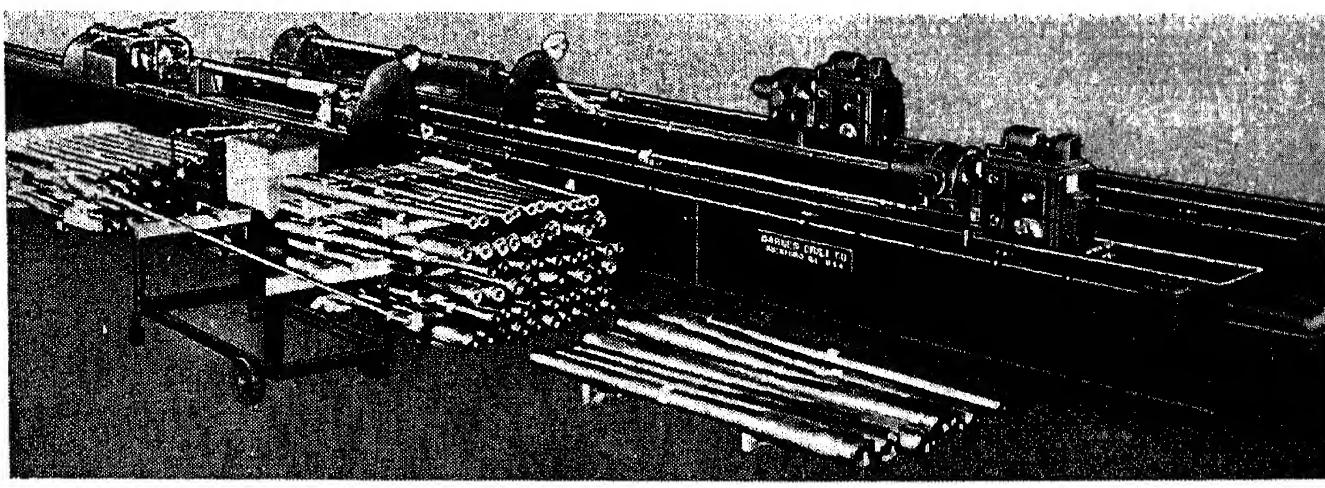


Fig. 19-5. A view of two No. 2 by 18 ft horizontal honing machines honing gun bores. (Courtesy Barnes Drill Co.)

spindle that rotates but is fixed lengthwise. A tool with a single stone extends from the front of the spindle. Different sizes of tools are necessary for different sizes of holes. Several spindle speeds are available. The operator places the work on the honing tool and depresses the foot pedal to start the spindle and expand the stone on the holder by means of a linkage passing through the spindle. The dial above the spindle indicates the size to which the tool is working at any moment.

On the *horizontal hydraulic honing machine* of Fig. 19-5, the workpiece is fastened in a horizontal position on the left. The honing tool is reciprocated and rotated by the head on the right that slides on the bed. The workpiece is rotated and the tool reciprocated on some machines. Honing machines of this type are made with strokes up to 75 feet and hone holes as large as 42 in. in diameter.

Vertical spindle honing machines hold the work and tools in a

position where gravity has the least effect upon the action. In general appearance, a vertical honing machine resembles a drill press. It may have a single spindle or a number of spindles, like the one in Fig. 19-6 for honing the cylinders of an engine block in production. The machine is capable of removing 0.004 in. of stock in each of 8 cylinders in 30 seconds and of holding size to within 0.0005 in.

Superfinishing

Purpose of Superfinishing. Superfinishing is a trade name given an operation using bonded abrasive stones under certain conditions to produce fine quality finished surfaces on metals. The process has been known since 1936 and has been found to give smooth, crystalline, and wear resisting surfaces on such parts as pistons, brake drums, bearing surfaces, bearing races, pump rods, valve tappet heads, valve stems, and piercing punches.

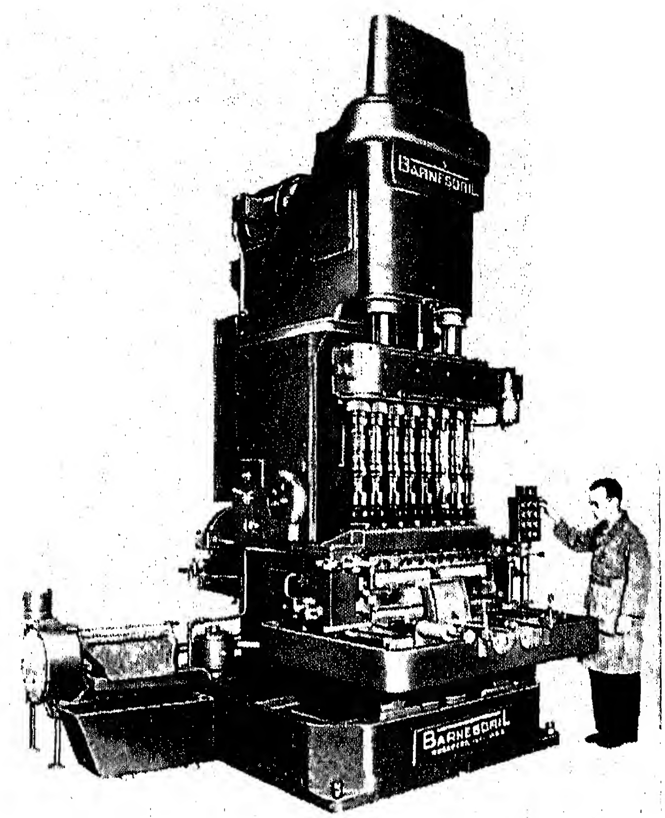


Fig. 19-6. A vertical multiple spindle hydraulic honing machine equipped with eight hydraulically actuated honing tools. An eight cylinder engine block is held in a fixture during the honing operation. (Courtesy Barnes Drill Co.)

Superfinishing is not essentially a dimension creating operation and removes on the average only from 0.0001 to 0.0002 in. of stock. Substantial geometrical and dimensional accuracy must be created first, usually by grinding. Superfinishing is intended to correct minute surface defects, is also effective in removing amorphous, fuzzy, broken, or burned material from the surface, and leaves a true base of parent metal in an accurate plane. Practically perfect surfaces with no apparent scratch pattern can be produced by Superfinishing. At the other extreme, Superfinished surfaces can be made with readings of 30 microinches rms and more and a deliberate crosshatched scratch pattern.

How Superfinishing is done. For cylindrical surface Superfinishing, a stone is pressed against a revolving workpiece and oscillated lengthwise. The pressure is from 10 to 40 lb per sq in., and the stone travels about 3/16 in. per stroke at 425 strokes per minute. The work generally revolves at 50 to 60 sfpm. A stone shorter than the workpiece is traversed to cover the entire work surface. The motions are arranged so that a grit never follows the same path more than once around the workpiece.

A 600 grit stone may be used for the smoothest surfaces, and coarser grits for definite scratch patterns. The stone has a soft bond, breaks down in use, and continually presents sharp abrasive grains. The stone is dressed nearly to match the curvature of the work surface and soon wears to a true fit. A stone normally has a width of 60 to 75 per cent of the workpiece diameter, is able to bridge and equalize a large number of surface defects at one time, and produces results reflecting the average form of the rough surface.

The workpiece and tool in Superfinishing are flooded with cutting fluid to carry away heat and particles of metal and abrasives. At first the stone touches only a few high spots, the pressure on them is high, and the cutting action is rapid. When the surface becomes smooth, the unit pressure decreases, and the stone rides on a film of fluid and ceases to cut. If no traverse is needed, a surface may be refined to 3 microinches rms or better in less than one minute.

Flat surfaces are Superfinished with an abrasive cup wheel on a vertical spindle with a spring loaded quill. The cupped end of the wheel bears on the surface of the workpiece carried on a lower vertical spindle. Both workpiece and wheel rotate but do not oscillate. Their centers are offset, and the wheel can be traversed radially on the workpiece. Quite true flat surfaces are produced when the spindles are exactly parallel, but they are not optically flat. If the wheel spindle is inclined, spherical surfaces are produced.

Superfinishing machines. Superfinishing may be done on a lathe with an attachment that mounts on the compound rest and presses and oscillates a stone on the work. However, machines specifically designed for Superfinishing are available for production. A general-purpose Superfinisher for cylindrical work is shown in Fig. 19-7. The workpiece is mounted and rotated between centers. The stone is carried by a head traversed and oscillated hydraulically

in the direction of the axis of the workpiece. Hydraulic pressure also pushes the stone against the work.

The machine of Fig. 19-7 can be provided with an attachment for flat parts, but other machines are specifically designed for flat surfaces. One of these has a vertical work spindle that rotates the work first under a roughing wheel spindle and then under a finishing spindle. A flat Superfinishing machine for large quantity production has nine spindles and is capable of Superfinishing 720 to 1200 valve tappet faces per hour.

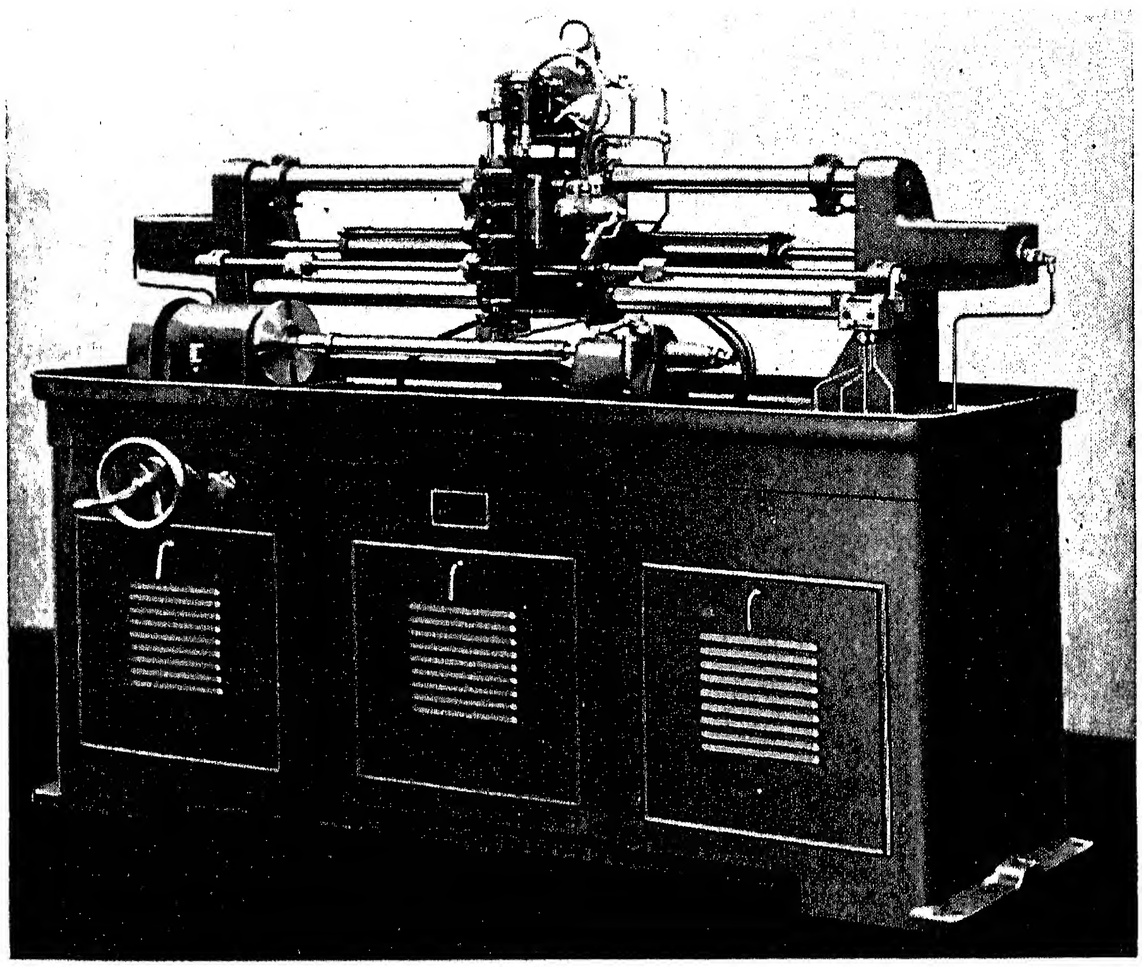


Fig. 19-7. A 4 in. by 36 in. general purpose cylindrical Superfinisher. (Courtesy Gisholt Machine Co.)

Polishing, Buffing, and Tumbling

Polishing removes scratches, tool marks, pits, and other irregularities from surfaces previously machined, cast, forged, etc. where accuracy of size or shape need not be controlled closely. Polishing

is done with abrasive coated polishing wheels or belts. The work may be applied by hand to wheels on floor-stand grinders like the one in Fig. 17-30. For production, semiautomatic and rotary polishing machines are available on which workpieces are placed on holders and fed to the wheels.

Buffing gives a high luster to a surface and generally follows polishing. Stock removal is small. The work is pressed against cloth or felt wheels or belts to which fine abrasive in a lubricant binder is applied from time to time.

Power brushing improves surface appearance and removes burrs and sharp edges, often in otherwise hard to reach places. Common power brushes are wire bristle wheels and Tampico wheels. The latter are made of tough fibers. Abrasive compounds may be applied to the brushes.

Tumbling and rolling clean castings, forgings, stampings, and screw machine products; remove burrs, fins, skin, scale, and sharp edges; take off paint and plating; improve surface finish and appearance; and have a tendency to relieve surface strains. Some reduction in size may be experienced. Workpieces to be treated are loaded in a barrel with abrasive particles, sawdust, wood chips, natural stones, cinders, sand, metal slugs, or other scouring agents depending upon the work and the action desired. Water is usually added, sometimes mixed with acids or other chemicals. The barrel is closed and rotated at a slow speed from one to more than ten hours, according to the amount of treatment desired.

Questions

1. What are the purposes of lapping, honing, and Superfinishing?
2. How is lapping done?
3. Describe three common forms of lapping machines.
4. How do lapping and honing differ?
5. How is honing done?
6. Describe the two general types of honing machines?
7. What does Superfinishing do and what does it not do?
8. How is Superfinishing done?
9. Describe several methods of improving surface finish where accuracy is not important.

References

Hemingway, E. L., *Wear and Surface Finish.* Madison, Wis.: The Gisholt Machine Co., 1947.

Manufacturers' Catalogs:

Barnes Drill Co., Rockford, Ill.

Cincinnati Grinders, Inc., Cincinnati, Ohio.

Gisholt Machine Co., The, Madison, Wis.

Micromatic Hone Corp., Detroit, Mich.

Norton Co., Worcester, Mass.

Sunnen Products Co., St. Louis, Mo.

Chapter 20

GEARS AND GEAR MAKING

A GEAR IS A MACHINE ELEMENT that transmits motion in a positive manner through teeth around its periphery. The *spur gear* is the simplest form with teeth parallel to its axis as shown in Fig. 20-1. A *rack* is a gear with an infinite radius, it moves in a straight line.

Gear tooth curves. The contact surfaces of gear teeth are curved to transmit motion uniformly as they roll and slide together. Two common curves that make good gear teeth are the cycloid and involute. The involute form is simple, efficient, easy to reproduce, allows variations in the center distances of mating gears, and has replaced the cycloid commercially. Sometimes an involute curve is modified at the extremities of a gear tooth and then is said to have a *composite* form.

An *involute curve* is generated by a point on a straight line rolling on a *base circle*. The line AB in Fig. 20-2 is tangent to the base circle and point A traces the involute profile of one side of the gear tooth as the line is rolled on the circle. In the position shown, the angle between the *line of action* AB and the tangent to the pitch circle is called the *pressure angle*. A *pitch circle* is an imaginary ring

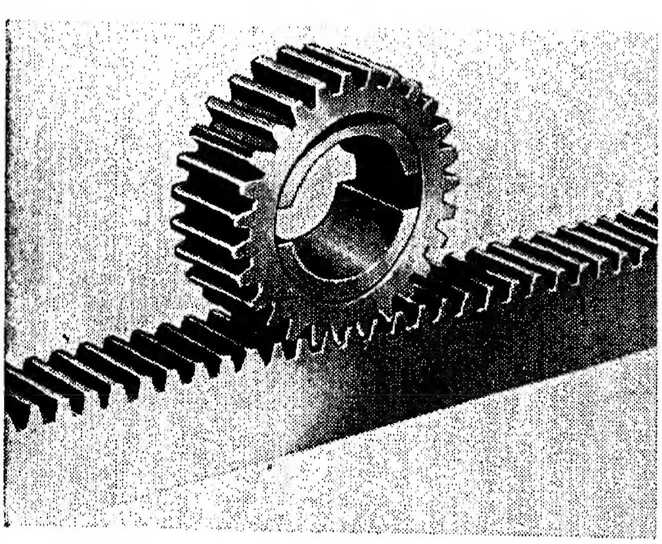


Fig. 20-1. A spur gear and rack. (Courtesy Foote Bros. Gear and Machine Co.)

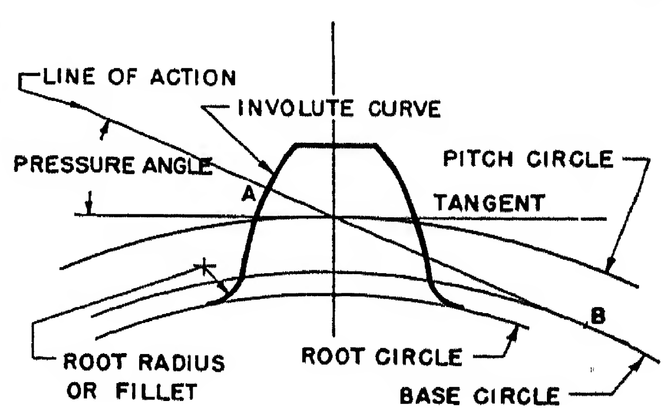


Fig. 20-2. An involute gear tooth.

of the same diameter as a smooth disk that would transmit the same relative motion by friction as the gear does when meshed with another gear. The involute curve obviously cannot exist inside of its base circle, and the tooth surface usually is radial there. For strength, the tooth surface is connected to the root by a radius or fillet.

Elements of gear teeth. Gear teeth could be made in an infinite number of sizes, thick or thin, long or short. However, if a series of gears is to mesh, their teeth must be uniform. On the other hand, gears and gear teeth must be made in a variety of sizes to do many jobs. Gear teeth have been standardized to satisfy these requirements. Two shapes of gear teeth are recognized. One is called the *full depth tooth* and is longer than comparable sizes of the

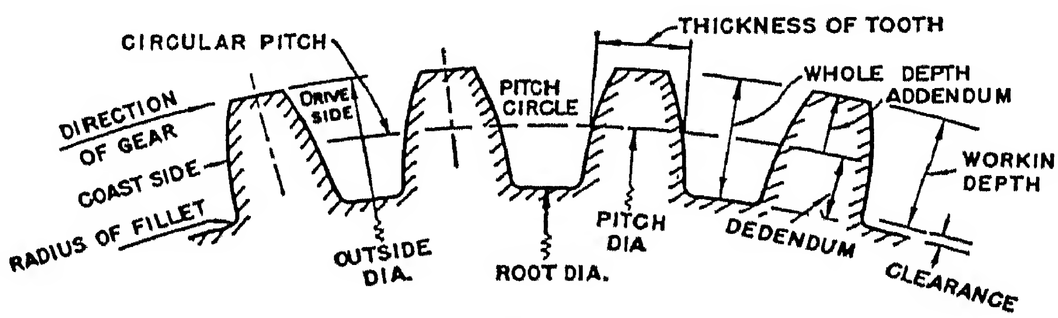


Fig. 20-3. The elements of gear teeth.

other, known as the *stub tooth.* Stub teeth are stronger but do not overlap as much as full depth teeth.

A gear tooth is identified by certain dimensions called elements. The common elements of gear teeth are designated in Fig. 20-3. The elements of a standard gear tooth are related to a factor called *diametral pitch,* which is defined as the number of teeth on the gear divided by the pitch diameter in inches. Any two standard gears of the same diametral pitch and tooth shape will mesh if mounted with the proper distance between their centers. The formulas for finding the dimensions of the main elements of full depth and stub gear teeth are given in Table VII.

Table VII shows that the circular pitch and tooth thickness at the pitch line are the same for all gears of the same diametral pitch whether they have full depth or stub teeth. On the other hand, the addendum, dedendum, and whole depth that determine tooth length are smaller for a stub tooth than for a full depth tooth of the same diametral pitch. Tooth size decreases as diametral pitch increases.

Table VII

Formulas for Calculating the Dimensions of Diametral Pitch Gears

Element name	*Symbol*	*Formula for:* *Full depth teeth*	*Stub teeth*
Addendum	a	$a = 1/P$	$a = 0.8/P$
Circular pitch	p	$p = \pi/P = \pi D/N$	$p = \pi/P = \pi D/N$
Clearance	c	$c = 0.157/P$	$c = 0.2/P$
Dedendum	b	$b = a + c = 1.157/P$	$b = a + c = 1/P$
Diametral pitch	P	$P = N/D$	$P = N/D$
Number of teeth	N	$N = P \times D = \pi D/p$	$N = P \times D = \pi D/p$
Outside diameter	D_o	$D_o = \frac{N + 2}{P} = D + 2a$	$D_o = \frac{N + 1.6}{P} = D + 2a$
Pitch diameter	D	$D = N/P = \frac{N \times p}{\pi}$	$D = N/P = \frac{N \times p}{\pi}$
Root diameter	D_R	$D_R = D_o - 2b_t = \frac{N - 2.314}{P}$	$D_R = D_o - 2b_t = \frac{N - 2}{P}$
Tooth thickness	t	$t = 1.5708/P$	$t = 1.5708/P$
Whole depth	b_t	$b_t = a + b = 2.157/P$	$b_t = a + b = 1.8/P$
Working depth	b_k	$b_k = b_t - c = 2/P$	$b_k = b_t - c = 1.6/P$

A full depth tooth of one diametral pitch is larger than a full depth tooth of two diametral pitch, which is larger than a full depth tooth of three diametral pitch, and so on.

Gear calculations. The important dimensions for making a gear are those needed to select the cutter, set the machine, prepare the blank, and inspect the finished gear. The dimensions most often needed for those purposes are the outside diameter, diametral pitch, number of teeth, tooth thickness, addendum, and whole depth. All may be given on the part print, but if the diametral pitch, number of teeth, and tooth width are specified, the other dimensions may be calculated for a standard gear. As an example, an eight diametral pitch full depth tooth gear has 32 teeth with a width of ¾ in. The width of the blank must be 0.750 in., and its outside diameter $\frac{32 + 2}{8} = 4.250$ in. Pitch diameter is $32/8 = 4.000$ in., tooth thickness is $1.5708/8 = 0.1964$ in., addendum is 0.125 in., and whole depth is $2.157/8 = 0.2696$ in. A stub tooth gear of the same diametral pitch and number and width of teeth would require a 0.750 in. wide blank with an outside diameter of $\frac{32 + 1.6}{8} = 4.200$ in. Its pitch diameter and tooth thickness are the same as for the full depth tooth, but for the stub tooth the addendum is $0.8/8 = 0.100$ in. and the whole depth is $1.8/8 = 0.225$ inch.

Spur gear tooth form systems. Various systems have been ad-

vocated for standardizing gear tooth forms and sizes. The older systems were sponsored by manufacturers of gears and gear-making equipment and bear such names as Brown and Sharpe, Fellows, and Maag. These have been succeeded by four systems proposed for spur gear tooth forms by the American Standards Association in a move to compromise and unify the independent systems. These four standard systems are:

1. A system that specifies a 14½° pressure angle full depth tooth with a composite form of tooth profile. The basic rack of this system does not have teeth with straight sides. Instead the profile is slightly curved at top and bottom. This system is almost the same as the older Brown and Sharpe System and is provided for gears that are form cut.
2. A system that specifies a 14½° pressure angle full depth tooth with a true involute form. The basic rack of this system has teeth with straight sides. It provides for gears cut by hobs and pinion-type shaping cutters. If a pinion and mating gear together have less than 64 teeth, the teeth of the mating gear will be undercut and weakened.
3. A system that specifies a 20° pressure angle full depth tooth with a true involute form and straight sided rack teeth. This system gives strong and quiet gears that can be generated, but the teeth may be undercut and weakened on small gears. An additional system has been sponsored by the American Standards Association for spur and helical gears of 20 diametral pitch and finer. This is called the 20 Degree Involute Fine Pitch System and provides a slight increase in whole depth to allow greater clearance.
4. A system that specifies a 20° pressure angle stub tooth with a true involute form and straight sided rack teeth. This system offers strong teeth and avoids undercutting.

The *Fellows 20° stub tooth system* has wide acceptance and deserves mention as an independent system. It designates diametral pitch by two numbers in the form of a fraction, like 6/8. The first number is the diametral pitch for calculating the tooth thickness, circular pitch, and number of teeth for a given pitch diameter. That is the number 6 in the example. The second number is the basis for calculating the addendum, dedendum, clearance, working depth, and whole depth. In this system the formula for dedendum is 1.250/P, and for the whole depth 2.250/P. Thus a gear of 6/8 dia-

metral pitch has a dedendum of 1.250/8 = 0.1563 in. and a whole depth of 2.250/8 = 0.2813 in. in the Fellows system.

Types of gears. Spur gears are the easiest and cheapest kind to make. They must be mounted on parallel shafts. A *helical gear*, like the one in Fig. 20-4, has teeth along a helix on a cylinder. The angle between the helix and an element of the pitch cylinder is

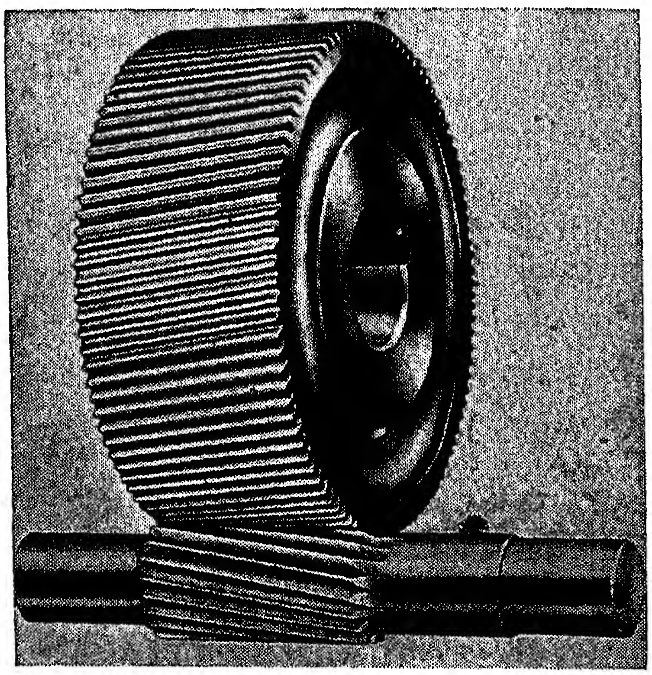


Fig. 20-4. A helical gear and pinion. (Courtesy Foote Bros. Gear and Machine Co.)

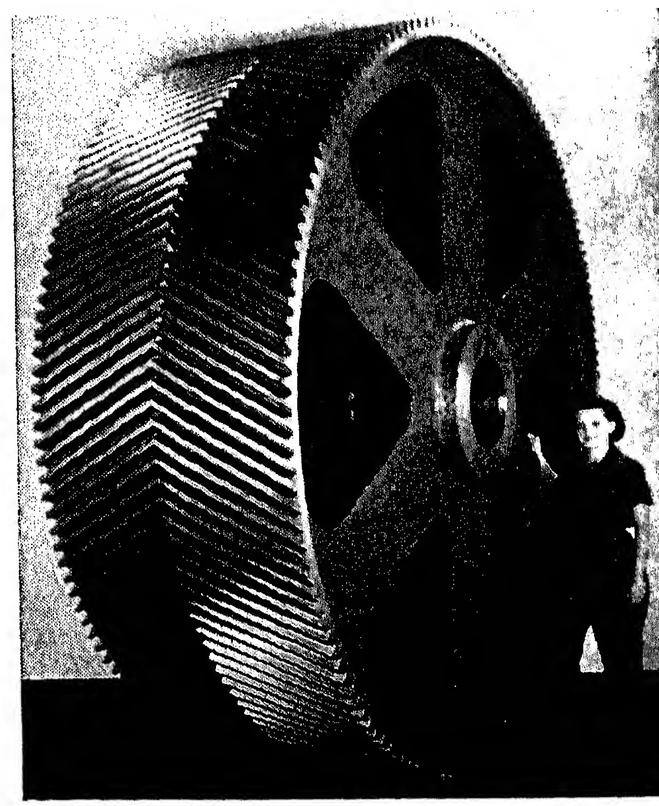


Fig. 20-5. A continous tooth herringbone gear. (Courtesy Foote Bros. Gear and Machine Co.)

called the *helix angle*. Helical gears are more expensive than spur gears but are stronger and quieter because the teeth engage gradually and more teeth are in mesh at the same time. They may be mounted on parallel shafts or on nonparallel and nonintersecting shafts.

A helical gear has a decided side thrust that is neutralized in a *herringbone gear*, like the one in Fig. 20-5, that has teeth on right- and left-hand helices. If the teeth come together in the center, they are said to be continuous, and the archlike construction makes a strong gear. A groove or gap around a herringbone gear to separate the teeth on two sides makes the gear easier to cut by some methods but weakens it.

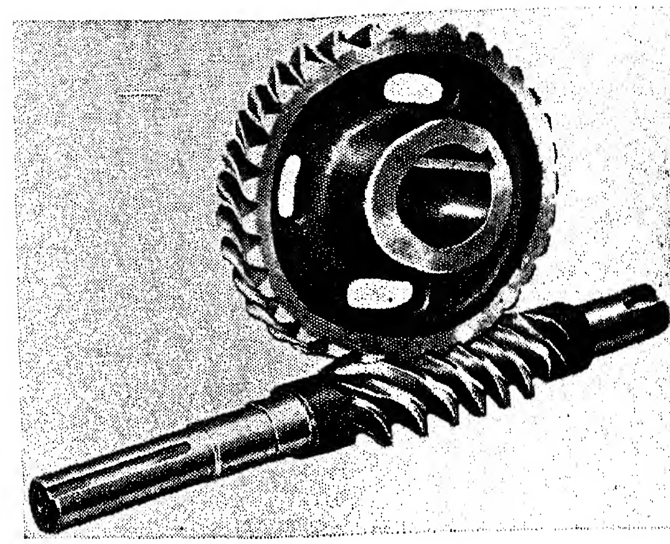


Fig. 20-6. A worm and worm gear. (Courtesy Foote Bros. Gear and Machine Co.)

A *worm* is like a screw and may have one or more threads, each a tooth. A high ratio can be obtained by engaging a worm with a *worm gear* having many teeth, as shown in Fig. 20-6. Their axes are nonintersecting and usually at right angles. A worm gear with a small helix angle cannot drive the worm, which is an advantage when a nonreversible drive is needed.

Bevel gears operate on axes that intersect at any required angle but most commonly at right angles. A bevel is conical in form. A *straight bevel gear* has straight teeth like the one on the machine in Fig. 20-13. If all the lines along its teeth were extended, they would pass through a common point called the *apex*. This apex point coincides with the point of intersection of the axes of mating bevel gears. A pair of bevel gears with equal numbers of teeth and perpendicular axes are called *miter gears*.

A *crown gear* is a bevel gear with a plane instead of a conical pitch surface. A crown gear is in the form of a disk and corresponds for bevel gears to the rack for spur gears.

The teeth of a *spiral bevel gear* are curved and oblique. One is shown on the left of Fig. 20-7. Spiral bevel gears run smoothly and quietly and are strong because their teeth have what is known as spiral overlap. They are relatively easy to manufacture. A *Zerol*

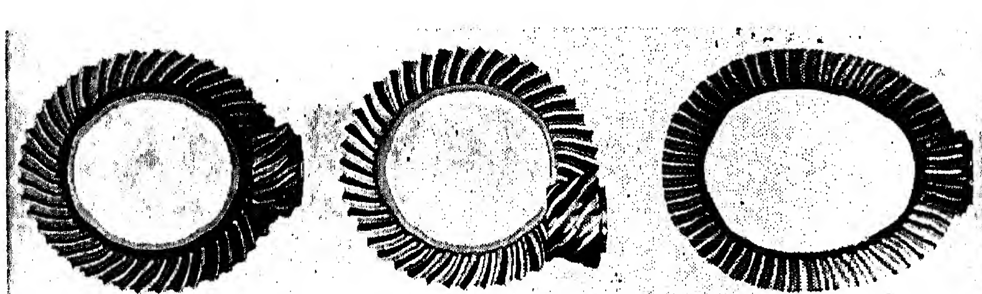


Fig. 20-7. A spiral bevel, hypoid, and Zerol bevel gear. (Courtesy Gleason Works.)

bevel gear has curved teeth, but they lie in the same general direction as straight teeth, as shown on the right of Fig. 20-7.

Hypoid gears resemble bevel gears but their axes do not intersect, as indicated in the middle view of Fig. 20-7. They are quiet and strong. A common application is for automobile rear axle drives.

The gears described so far are all *external gears.* An *internal gear* is one with teeth inside a cylinder or cone. Internal gears are used for clutches, speed train reducers, and planetary gear trains.

Methods of making gears. Gears are produced by the four general methods of casting, molding, hot rolling, and machining.

Gears may be cast in sand or in permanent molds. Cast iron gears are rough, inaccurate, and low in strength, but large sizes can be made at relatively low cost. Many small gears for light service are die cast of zinc, tin, aluminum, and copper alloys. They can be made to a high degree of accuracy and finish.

Gears are sometimes molded of plastic materials where quietness and insulating properties are needed and only moderate strength for light service is required. Gear stock of brass or aluminum is extruded, to be cut to desired lengths. Some gears are pressed and sintered from metallic powders.

Gear rolling has been done by rolling a hot blank with a master gear as the two are brought together, but the method has not been generally accepted.

Gears are machined from cast and forged blanks, bar stock, sheet metal, laminated plastics, and molded shapes. Gears stamped from sheet metal are found in watches, clocks, toys, and many appliances. Gears of all grades are machined from solid stock. Accurate and hard gears for severe service are produced by precise finishing methods.

Gear cutting methods may be divided into three classes as follows:

1. The *forming* method, which uses a cutter having the same form as the space between the teeth being cut. The cutter may be a single point tool on a planer or shaper, a rotating cutter on a milling machine, or a broach.
2. The *template* method, in which a cutting tool is guided by a master former or template on a machine called a gear

planer. This method is suited for large and coarse pitch bevel and spur gears.

3. The *generating* method, in which the cutting profile of the tool is like that of a mating gear or rack tooth. The cutter and work roll together as though in mesh to develop the tooth form.

Form Cutting

Cutting gear teeth on a milling machine. Spur, helical, worm, and straight bevel gears may be cut on milling machines with standard dividing heads and arbors. Gear cutters are the only tools needed that are not used for other kinds of operations and their cost is low. Setup is easy. At one time most gears were form cut but during the present century generating has proven to be a more efficient method for manufacturing gears. Milling machines are not used in modern gear manufacture but only when one or a few gears are made at a time and when more efficient equipment is not available. That is because the cutting of gears on a milling machine is a relatively slow and inaccurate process.

A dividing head or similar indexing device is mounted on the table of the milling machine at right angles to the machine spindle for cutting a spur gear. The dividing head is arranged to index the required number of teeth. The work usually is placed between centers, often on a mandrel, and connected by a dog to the dividing head spindle. The cutter is mounted on an arbor in the machine spindle and is centered with reference to the point of the dividing head center. Some gear tooth cutters have a central line for that purpose. Speeds and feeds are selected, and the work is positioned under and just touching the cutter. The table is then moved to starting position to clear the cutter, and the knee is raised a distance equal to the whole depth of the gear being cut. The knee is clamped to the column, and the saddle to the knee. The cutter travels across the gear blank and is returned to starting position after each pass, and the work is indexed one tooth space. After two spaces have been cut, the tooth thickness is checked, and further adjustments are made if needed.

A universal milling machine or universal spiral milling attach-

ment is required for milling a helical gear or worm so that the cutter can be aligned with the tooth space at the proper helix angle. Typical setups are shown in Figs. 12-2 and 13-2. The dividing head is geared to the table leadscrew through a helical milling attachment. The workpiece is indexed by hand for each pass.

A rack is milled with a rack milling attachment as shown in Fig. 13-4.

Bevel gears with theoretically correct tooth forms cannot be cut with rotary cutters on a milling machine. The teeth must be filed by hand after being cut to make them perform satisfactorily. A bevel gear blank is chucked on a universal dividing head and inclined at an angle equal to its root cone angle. The gear is moved off center and rolled back toward the cutter to finish one side of all the teeth. It is then moved and rolled in opposite directions to finish the other side of the teeth. Instructions for calculating offset and rollback are given in handbooks.

Gear tooth form cutters. Commercial form relieved cutters for spur gears with a 14½° pressure angle and composite form are available in sets. A set for each diametral pitch contains 8 cutters numbered from 1 to 8. Each cutter covers a range of certain numbers of teeth and is only an approximation for all but one gear. For instance, number 8 is for gears having 12 and 13 teeth, and number 1 for 135 teeth to a rack. For more accuracy, intermediate cutters numbered 1½ to 7½ are available. A *stocking cutter* is for heavy rough cuts and has grooves on the sides of its teeth to break up chips. Gear tooth cutters appear in Fig. 12-16.

A spur gear cutter for a certain pitch and number of teeth is not suitable for cutting a helical gear of the same number of teeth. A different cutter must be selected to correspond to the tooth form across the direction of cut.

Cutters for bevel gears are similar to those for spur gears but are thinner in order to pass between the teeth at the small ends of the gears. They are made in sets and stamped with the word "bevel." Manufacturers' catalogs give instructions for selecting them.

Production form cutting. Gears sometimes are roughed out by form cutting and then finished by generating when manufactured in quantities. Form cutting machines are made for the specific purpose of roughing gears rapidly. One type uses a circular form

cutter and operates like a milling machine, but has a built in indexing mechanism and roughs out gears automatically. The operator has only to unload and load the work. Such machines are also commonly used for cutting slots and grooves around parts other than gears.

A high production machine called the *Shear Speed Gear Shaper* form cuts spur and helical gears preparatory to finish shaving. It is a fast machine and can turn out gears up to 10 in. diameter by 2¾ in. wide in 13 to 50 seconds each. A number of single point cutters are arranged in a circle in a hollow vertical head. Each cutter has the form of a tooth space. The gear blank is mounted on a fixture below the head and is reciprocated into the head. The tools are fed in uniformly at each stroke of the work and are retracted when they have cut to depth. All the tooth spaces are cut at the same time.

Generating Spur and Helical Gears

Gear Hobs. Hobbing is a generating process done with a cutter called a *hob* that revolves and cuts like a milling cutter. Its teeth lie on a helix like a worm. A typical hob is illustrated and its elements are designated in Fig. 20-8. Lengthwise gashes expose the

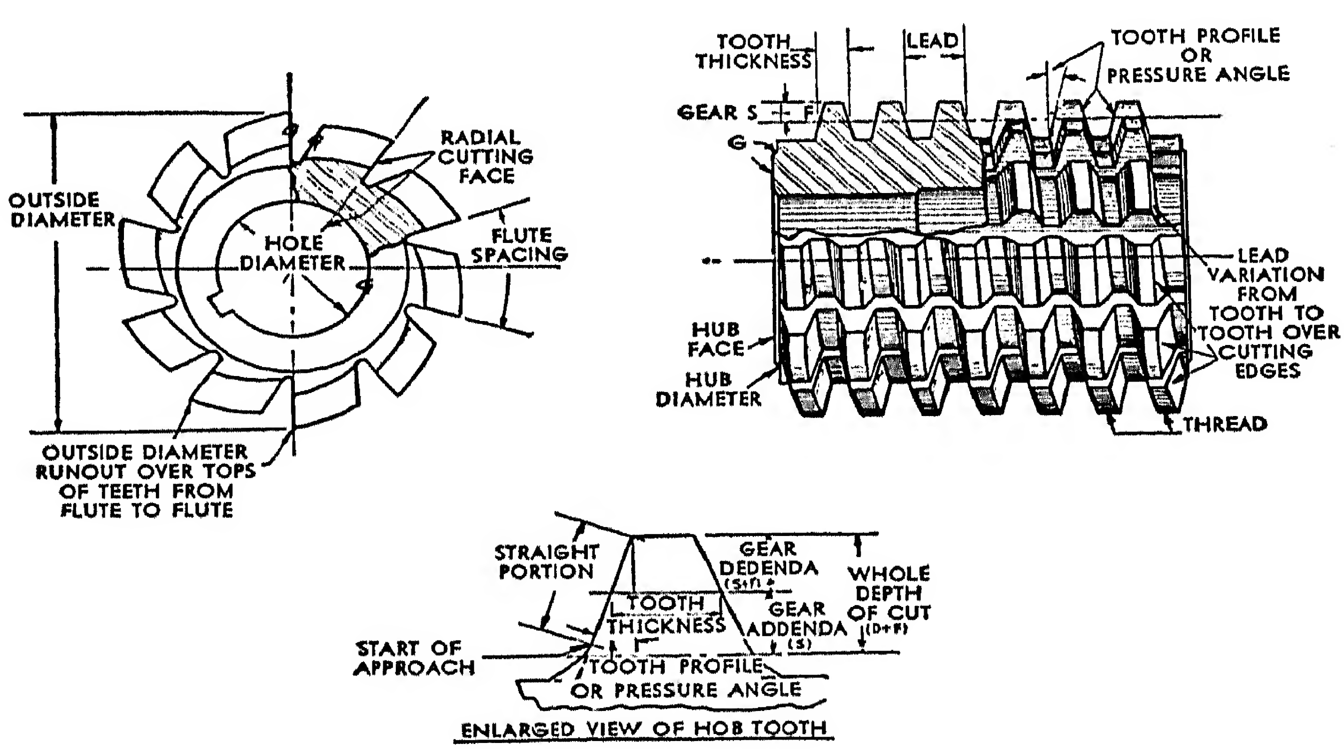


Fig. 20-8. A hob and its elements. (Courtesy Illinois Tool Works Co.)

cutting faces that have a contour simulating a rack. The teeth are form relieved behind the cutting edges. As the hob revolves one turn, the effect is as though the simulated rack were to move lengthwise an amount equal to the lead of the thread on which the teeth lie. When a gear is cut, it is positioned and revolved as if it were in mesh with such a rack. A hob cuts all gears having the same tooth form and diametral pitch as the rack it represents.

A hob may have one, two, or more threads. A gear being cut by a single thread hob must turn an amount equal to its circular pitch for each revolution of the hob. The amount is twice the circular pitch of the gear for a double thread hob, and so on. Thus the speed ratio between a gear and hob depends upon the number of teeth on the gear and the number of threads on the hob. For example, a single thread hob rotates 30 times for each revolution of a 30 teeth gear, but a double thread hob rotates only 15 times for each revolution of the same gear. A single thread hob does not produce at as high a rate as a multiple thread hob but cuts a more accurate gear.

Hardened but unground hobs are satisfactory for average work, especially for roughing. Hobs are ground all over after hardening for accuracy. Hobs are made straight, tapered, and formed. A straight hob is shown in Fig. 20-8 and is the most common. A heavy load can be distributed among more teeth with a tapered hob. When a straight hob is cutting, a few of its teeth become dull first. The hob is then moved over so that sharp teeth carry the load. That is repeated several times before the hob is resharpened.

A hob for cutting a worm gear is preferably a counterpart of the worm that is to engage the gear.

Hobbing machines. A gear on a horizontal work spindle is being hobbed on a hobbing machine in Fig. 20-9. The hob is carried on a spindle in a housing that can be swiveled on a carriage. The angle of swivel is equal to the lead angle of the hob to cut a spur gear. For a helical gear, the angle of swivel is equal to the helix angle of the gear plus or minus the lead angle of the hob, depending upon the hands of the helices. The hob carriage slides on ways along the bed of the machine to feed the hob across the workpiece.

The gear being cut in Fig. 20-9 is held on an arbor revolved by the spindle in the workhead which is carried on a column rising from the bed. A heavy overarm extending from the top of the work-

head carries a tailstock supporting the outer end of the work arbor. The outer end of the overarm is clamped to an outboard support fastened to the end of the machine bed. The workhead is raised, lowered, and positioned by means of a vertical screw and micrometer dial to adjust the work for the hob to cut to the correct tooth depth.

A constant speed motor drives the hob through speed change

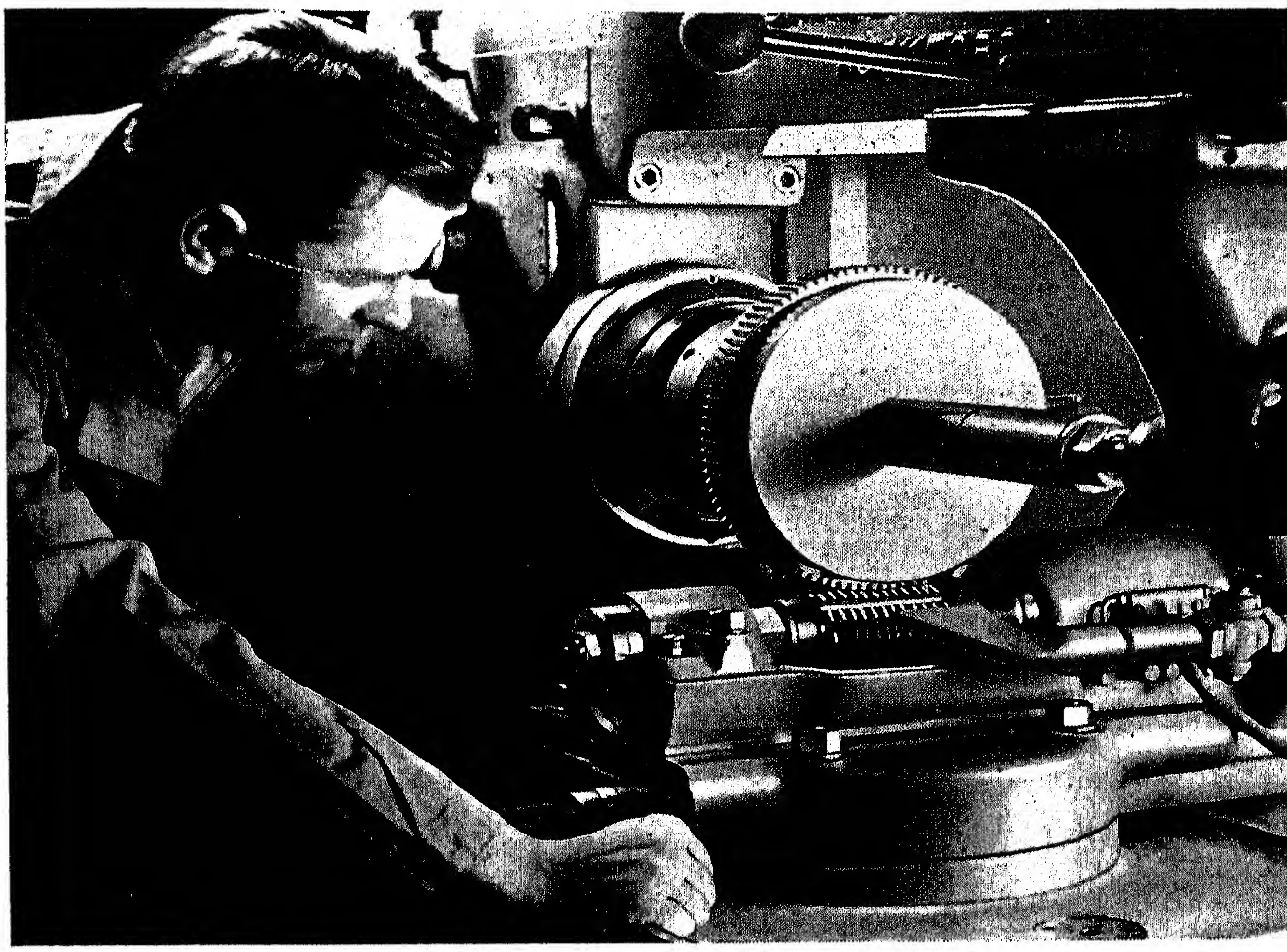


Fig. 20-9. A gear being hobbed on a hobbing machine. (Courtesy Barber Colman Co.)

gears which are selected to make the hob rotate at the proper surface speed, like a milling cutter. The hob spindle and work spindle are connected by a positive drive through index change gears that are selected to give the proper ratio of work speed to hob speed. The feed of the hob across the gear is expressed in inches per revolution of the gear. This feed rate is obtained from feed change gears in a drive from the work spindle to the leadscrew that moves the hob carriage.

Some hobbing machines are equipped with a differential mechanism in addition to that already described for ordinary or nondifferential machines. The differential mechanism is advantageous for cutting helical gears because the relation of the hob to the teeth of the gear is automatically maintained even though the feed is disengaged, which is not possible with nondifferential hobbing. A differential hobbing machine is more complex and expensive than one without a differential.

Hobbing machines are made in many sizes and styles from small ones for watch and instrument gears to huge ones for gears over 10

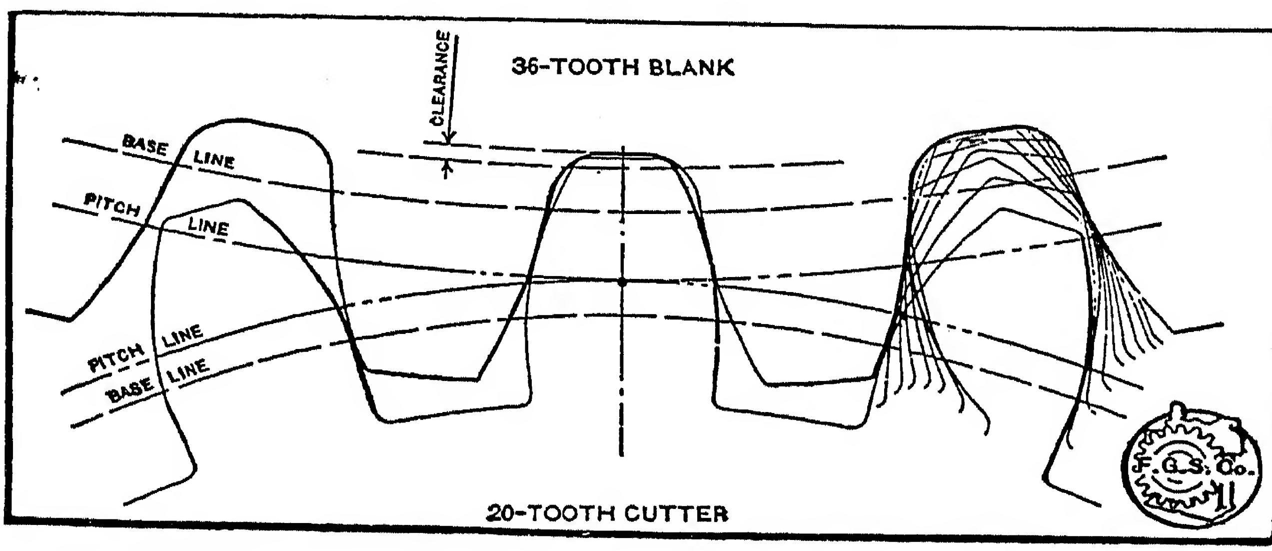


Fig. 20-10. The generating action of a rotary gear shaper cutter. (Courtesy The Fellows Gear Shaper Co.)

feet in diameter. Some are general-purpose machines, others are specialized. In addition to the horizontal type, hobbing machines are made with vertical work spindles. They are convenient for large gears. For large quantity production, hobbing machines are made with several work stations and often are arranged for automatic operation.

Gear shaping. A gear is shaped by a reciprocating cutter in the form of a single tooth, rack, or pinion. The gear blank is revolved as though it were in mesh with the cutter. A series of cuts taken by a pinion type cutter is depicted in Fig. 20-10. A shaper cutter is capable of generating any gear of the same pitch.

The teeth of the typical pinion-type gear shaper cutter of Fig. 20-11 have a true involute form at any section parallel to the face and are form relieved on the sides, outside, and root for clearance.

The face of the cutter is dished to provide rake. As this face is ground to resharpen the cutter, the teeth become smaller but retain their involute form and pitch. Cutters for helical gears have helical teeth. This cutter has a hole in the center for mounting on a stub arbor or machine spindle. Small cutters have tapered shanks.

Gear shapers. The gear shaper in Fig. 20-12 can cut gears with pitch diameters up to 18 in. and widths up to about 6 in. Lighter models are made for smaller gears and operate faster. The cutter

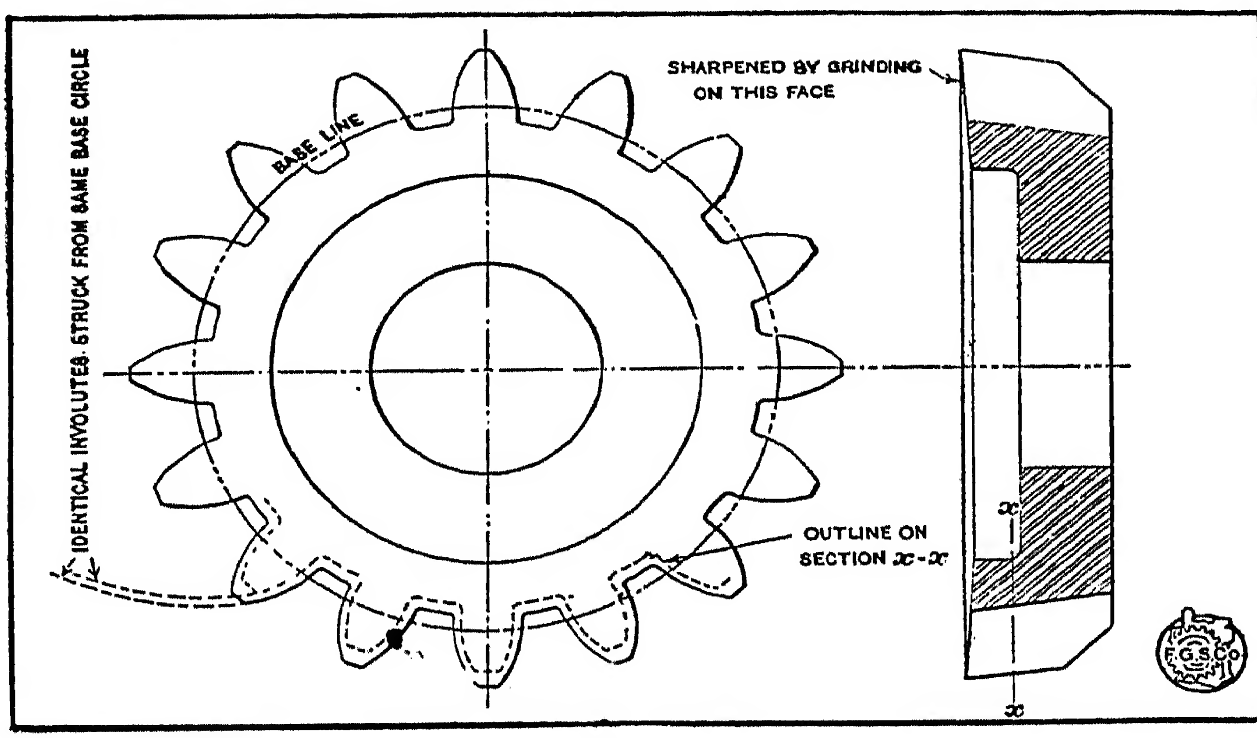


Fig. 20-11. A pinion type gear shaper cutter. (Courtesy The Fellows Gear Shaper Co.)

is held on the lower end of a reciprocating spindle in the head that slides on ways on the body of the machine. For helical gears, guides are inserted in the head to give a twist to the cutter spindle as it reciprocates up and down. The cutter is revolved as it is reciprocated. A choice of five rates of strokes per minute and six numbers of strokes per revolution of the cutter is given. The length and position of stroke can be adjusted to suit the gear being cut.

The work is mounted on a vertical spindle below and to the right of the cutter spindle and is retracted from the cutter on the upstroke. Change gears are put in the drive between cutter and work spindles to rotate the workpiece in time with the cutter.

The cutter is moved aside when the work is loaded on a gear

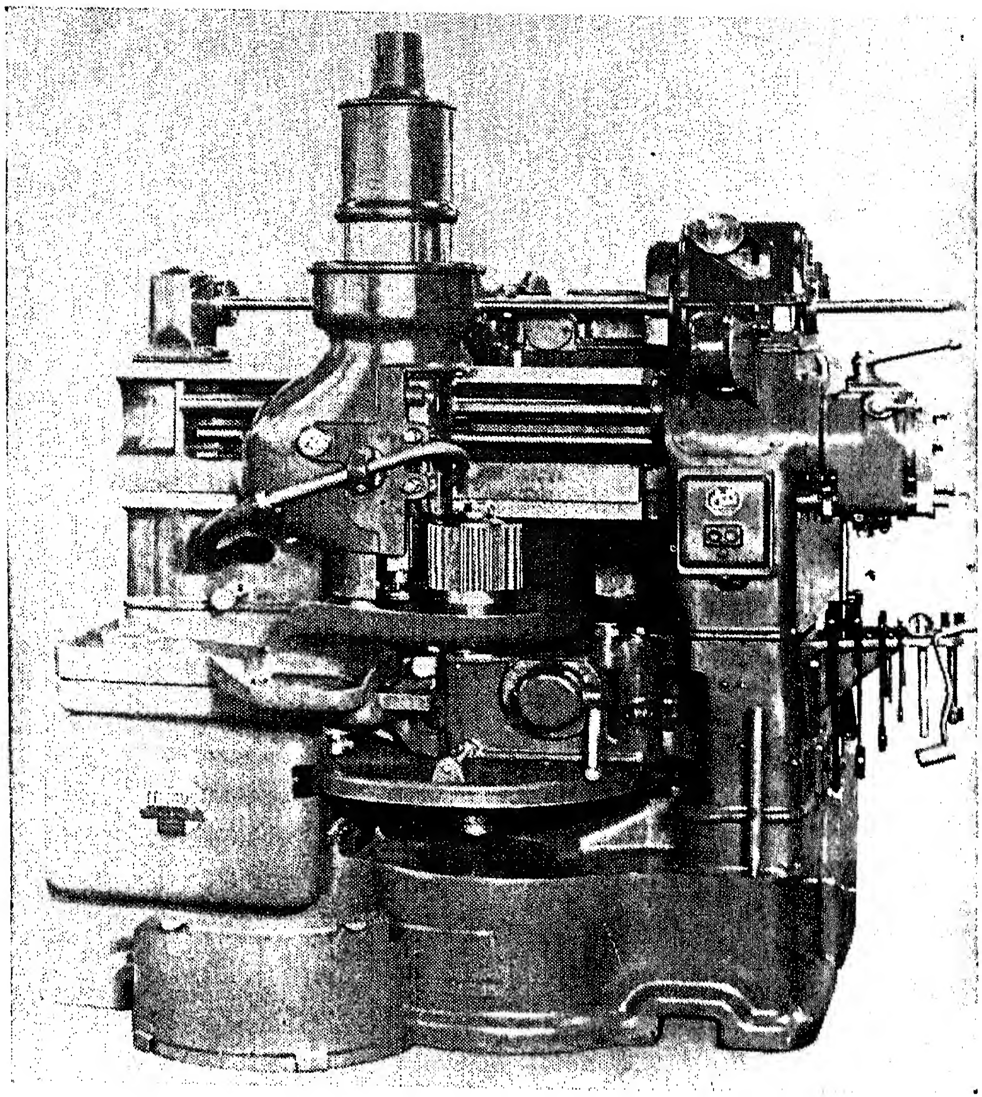


Fig. 20-12. An 6A type Fellows gear shaper. (Courtesy The Fellows Gear Shaper Co.)

shaper. When the machine is started, the cutter is fed radially to depth while revolving with the workpiece. After the cutter has reached the desired depth, the cutting action continues for at least a full revolution of the gear. Two cuts around a gear are necessary for best results. Both may be taken on one machine, in which case the workpiece is turned through two revolutions. Economical re- results are obtained in production by taking the roughing cut on one machine and the finishing cut on a second machine.

Vertical gear shapers are made with multiple work stations for large-quantity production. One machine has 10 stations mounted around a central rotary drum. The stations are unloaded and loaded in turn as they pass the operation's position. Another high production gear shaper carries a 16 in. diameter reciprocating cutter

around which blanks in six stations revolve in a planetary fashion. The workpieces are unloaded when they reach a gap in the cutter.

The gear shapers described so far utilize pinion-shaped cutters. On gear shapers with rack-type cutters, the workpiece rolls along the reciprocating rack. If the rack is shorter than the distance around the gear, the action is repeated in stages until the entire gear is cut. A small machine of this type for meter, camera, and clock gears is entirely automatic with mechanical means for loading and unloading the work. Larger machines are available for gears up to several feet in diameter.

The *Sykes gear generator* has two pinion-type reciprocating cutters on horizontal in line opposed spindles that work alternately. The work is also carried on a horizontal spindle. This gear shaper cuts gears up to 12 ft in diameter, including continuous tooth herringbone gears.

A *rack shaper* is arranged to cut racks instead of gears with a pinion-type cutter that is revolved as it is reciprocated. The machine resembles a vertical gear shaper but has a long table on which the work is mounted and fed tangentially past the cutter. A typical rack shaper can cut racks up to 72 in. long with a 4 in. tooth width.

Comparison of gear hobbing and shaping. Spur, helical, and worm gears, worms, ratchet wheels, and sprockets for chain drives are produced by hobbing and shaping. In addition, the processes are capable of machining a variety of other shapes including straight tooth and involute splines, square and hexagonal shafts, and cams.

Except for the methods described for production form cutting, gear generating methods are faster and more accurate than form cutting. Many gears semifinished by hobbing and shaping are finished by burnishing, shaving, or grinding, which are described later.

About 70 per cent of all cut gears are hobbed. The continuous action of the hobbing process makes it generally faster and more accurate than competing processes. The heat generated in hobbing is dispersed uniformly over the workpiece and cutter. The nature of hobbing requires a relatively simple machine with few motions, but a hobbing machine must be rugged because of the varying action of the hob under cut. Fairly long shafts, splines, or a batch of gears on one arbor can be accommodated on most hobbing machines. Herringbone gears of the gap type only can be hobbed.

Gear shaping is applicable to cutting internal gears, gears close to a flange, cluster gears, and continuous herringbone gears that cannot be cut by rotating cutters. Among other products of the gear shaper are interrupted tooth gears, elliptical gears, face gears, racks, cams, and pawls. The width of gear tooth that can be cut is limited on many gear shapers, but on the Sykes gear shaper two members of a cluster gear or gangs of gears can be cut at one time.

Rack-type gear cutters are easiest to make accurately because their teeth have straight sides. Hobs are next easiest to make, and pinion-type shaper cutters are the most difficult.

Bevel Gear Cutting*

Machines for cutting bevel gears may be divided into two classes, (1) for straight and (2) for curved teeth. The basic machines of universal type are the two-tool straight bevel generator and the spiral bevel and hypoid generator. Others are available for special or supplementary purposes, and the most important of them will be described briefly.

Machines and methods for straight tooth bevel gears. A view of the cutting tools and a gear in position on a two-tool *straight bevel generator* is shown in Fig. 20-13. Two tools are used, one on each side of a tooth, to make the tooth taper in the desired manner. They reciprocate along slides on a cradle, and their tips travel along paths directed through the apex of the gear being cut. The tools have straight cutting edges and simulate the sides of a tooth space of an imaginary crown gear. The cradle and tools roll upward with the workpiece as though the simulated crown gear were in mesh with the gear being cut. During the roll, the tools make a series of cuts to develop the tooth shape in a manner similar to that demonstrated for shaping spur gears. A gear with any number of teeth within the capacity of the machine may be cut by arranging the proper relative motion between the tools and blank. One tooth is generated from the bottom to the top of the roll. The gear is then withdrawn and indexed. The cradle is returned to the bottom position and the gear is fed into full depth with the cutting tools. Another tooth is cut during the next roll, and so on.

* This section reviewed and revised by Gleason Works, Rochester, N. Y.

A generator for small straight tooth bevel gears employs two disk-type milling cutters with interlocking teeth instead of the conventional reciprocating tools. The cutters are rolled with the blank to generate the teeth. Straight bevel gear generators are available in several sizes for gears from about 3/16 in. to 35½ in. diameter.

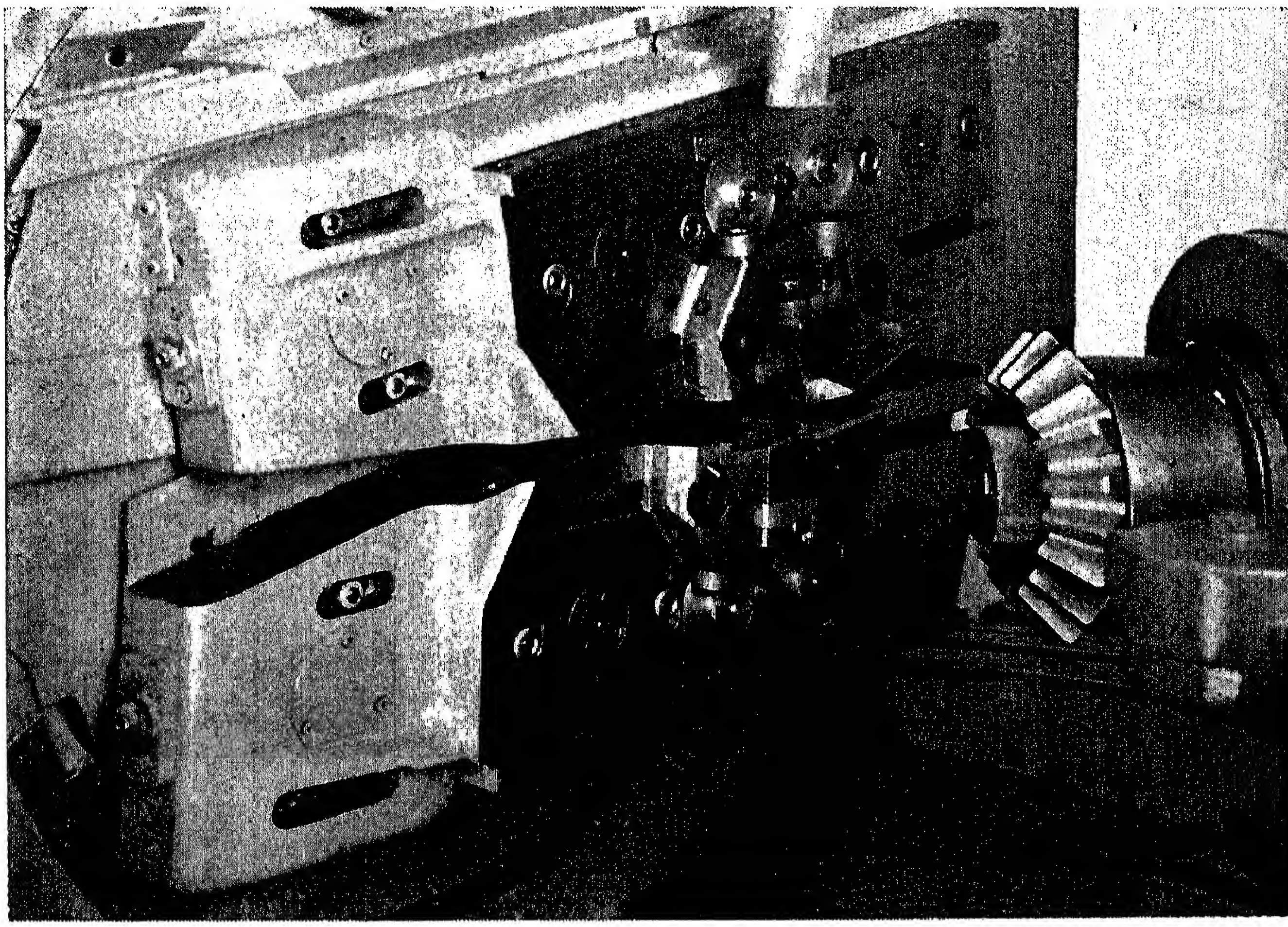


Fig. 20-13. Straight tooth bevel gear and generating tools on a straight bevel gear generator. (Courtesy Gleason Works.)

On the latest types of straight bevel generators, the teeth can be slightly crowned from end to end to localize the tooth contact. A gear with crowned teeth may be displaced slightly in assembly or under load without the load's being concentrated at the ends of the teeth where it is dangerous. The same effect is obtained from a difference in curvature of the mating surfaces of bevel gears with curved teeth.

Fine pitch straight bevel gears are mostly cut from the solid in one operation. Certain methods combine roughing and finishing in one operation, even for fairly large teeth. Otherwise, all straight bevel gears must be rough cut prior to finish cutting. They may be

roughed in small quantities on the two-tool straight bevel generator without generating motion. The tools are similar to finishing tools but are ground with rake suitable for roughing. Machines and tools especially designed for rapid roughing without generation are commonly used to prepare bevel gears for finishing when quantities are large.

The fastest known cutting process for straight bevel gears is provided by the *Revacycle Process* for high production. The rotating Revacycle cutter roughs and finishes a tooth space during each revolution of the cutter. Automatic loading and handling devices often are added to the machines operating by this high production method.

Straight bevel gears too large for generating machines and large spur gears are cut on the *gear planer*. This is the oldest type of machine capable of cutting bevel gears with teeth tapering in the correct manner. A single planing tool with rounded point is re-

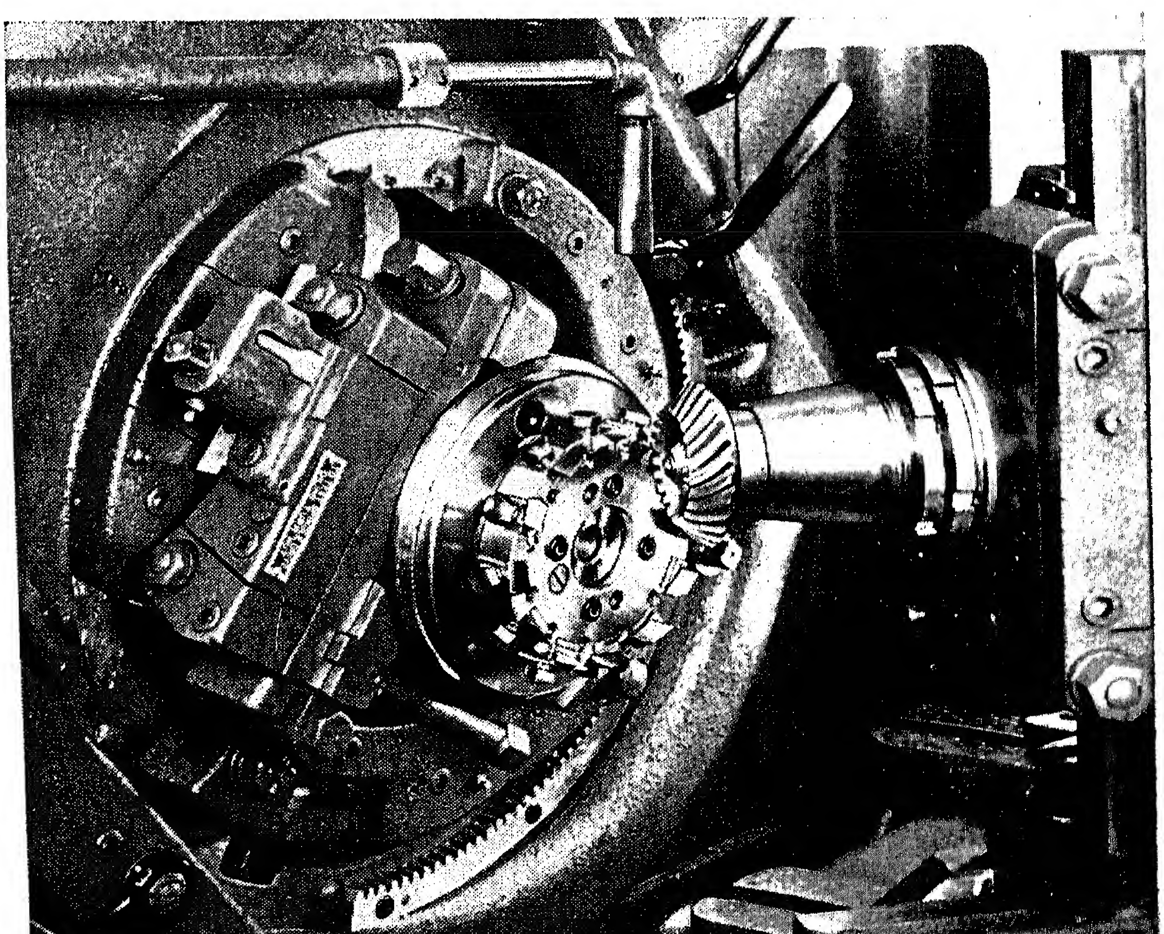


Fig. 20-14. Spiral bevel gear, rotating cutter, and cradle on Gleason Hypoid Generator. (Courtesy Gleason Works.)

ciprocated across the face of a gear and is controlled by a template or former to produce the profile shapes of the teeth.

Machines for curved tooth bevel and hypoid gears. A *spiral bevel gear and hypoid generator* employs a special form of face mill like the one on the machine of Fig. 20-14. The cutter represents a tooth of a crown gear and is rolled with the gear being cut to generate the tooth profile. The machine can be adjusted for spiral bevel, Zerol, and hypoid gears. Sometimes both sides of a tooth space are finish cut in one operation, and sometimes only one side, depending upon the tools available or the quality and quantity of gears required. Machines and cutters are available for gears from $\frac{3}{16}$ in. to 33 in. diameter.

Spiral bevel gears, except those of fine pitch, are usually first rough cut and then finish cut. *Spiral bevel roughers* rough cut gears without a generating motion and pinions with a generating motion.

For large quantities of spiral bevel and hypoid gears of ratios of 3 to 1 and larger, the larger member of a pair is often finish cut without generation on a machine especially designed for that purpose. This decreases the cutting time and cost. The pinion is generated to suit the gear. Such pairs are called *Formate gears.*

Spiral bevel, Zerol, and hypoid gears too large for generators with rotating cutters are cut on the *planing generator*. It employs a single planing tool that cuts tooth after tooth around the blank that rotates continuously. The tool is carried on a cradle that is rolled with respect to the workpiece to provide the generating action.

Spiral bevel grinders are available in both generating and Formate types. They finish grind the teeth after the gears have been semifinished and hardened. This corrects inaccuracies caused by hardening and promotes uniformity in the gears.

Gear Finishing

A gear tooth surface that is hobbed or shaped is composed of tiny flats. Such a surface is satisfactory for some purposes but is not good enough where a high degree of accuracy and stamina is required. The flats can be made very small by cutting at low feeds, but the operation then becomes slow and costly. Often the lowest net cost

can be realized by cutting a gear fast with less accurate and less expensive tools, and then adding a finishing operation.

Tooth straightness, size, concentricity, and spacing, and involute form are difficult to control within the closest limits in the gear cutting processes in which appreciable amounts of material are removed. This is the same condition that prevails in all metal machining operations. As a rule rough cuts must be taken to remove large amounts of stock and must be followed by finishing cuts for good surface finish and accuracy. Furthermore, gears often are heat treated for hardness and strength, and that tends to warp them and form scale.

Errors in gears make them noisy. That may be quite objectionable in, say, an automobile transmission. Gears that must transmit motion accurately must be accurate themselves. Such gears are found in fire control instruments and timepieces. Gears that are heavily loaded, like those in aircraft engines, will fail if their teeth vary so that some are overstressed while others do not take their full share of the load. Gear finishing operations are performed to make gears quiet, smooth running, and dependable. They include shaving and burnishing for soft gears and grinding and lapping for hard gears. Only a few thousandths of an inch or less of stock are left on gear teeth for finishing.

Gear shaving. A gear is shaved by running it at high speeds in mesh with a cutter in the form of a rack or gear with gashes or grooves on its tooth faces, as typified in Fig. 20-15. The edges of these grooves are sharp and actually scrape fine chips from the faces of the teeth of the workpiece. Some sliding normally takes place between gear teeth in mesh, and this action is augmented in gear shaving by crossing the axes of the cutter and work gear and by reciprocating the work as it is rolled with the shaving cutter. Except for shaving a gear close to a shoulder, the nonintersecting axes generally cross at an angle of 10° to 15°. The tooth form of the cutter is accurately ground and reproduces a correspondingly accurate conjugate form on the workpiece. Shaving can be made to crown gear teeth slightly at their centers to localize tooth contact and keep it away from the ends of the teeth.

A typical *rotary shaving machine* for medium-size gears is shown in Fig. 20-16. The workpiece is mounted on an arbor between live centers on a reciprocating table. The helical shaving cutter drives

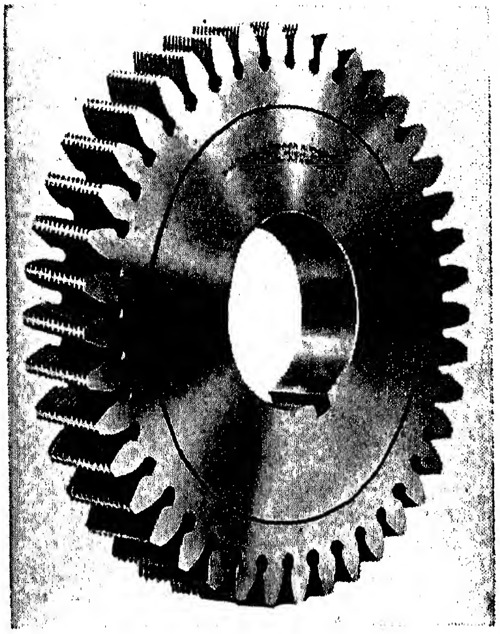


Fig. 20-15. A rotary gear shaving tool. (Courtesy National Broach and Machine Co.)

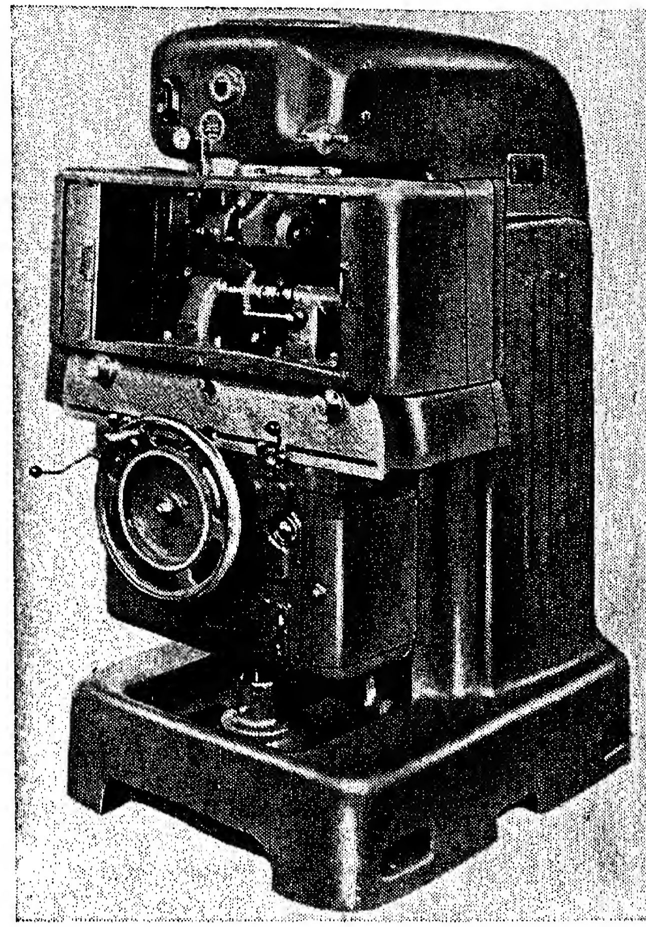


Fig. 20-16. A 12 in. rotary gear shaving machine. (Courtesy National Broach and Machine Co.)

the gear and is mounted above the work in a head swiveled at an angle with the gear axis. The knee that carries the table is raised to force the gear against the cutter and lowered to unload the work. The feed and traverse are automatic, and the duration of a shaving cycle is controlled by a timer.

A rack-type shaving cutter is reciprocated lengthwise at high speed under and in mesh with a workpiece on a *rack-type shaving machine.* The workpiece is reciprocated sideways and fed into the rack.

Gear shaving is a low cost rapid production process. Most gears can be shaved in less than half a minute apiece, some in as short a time as five seconds. Each cutter is suitable only for a single pitch and tooth form. Cutters are expensive but in most cases are capable of shaving thousands of gears before having to be resharpened. Gear shaving machines are made in many sizes, to finish gears from the smallest up to 10 feet in diameter.

Gear burnishing. A gear is mounted on a burnishing machine on an upright floating spindle in mesh with three hardened burnish-

ing gears, one of which is power driven. The burnishing gears are forced inward against the work gear and turn a few revolutions in each direction. The tooth surfaces of the work gear are smoothed and slightly hardened but are left with a layer of weak smear metal.

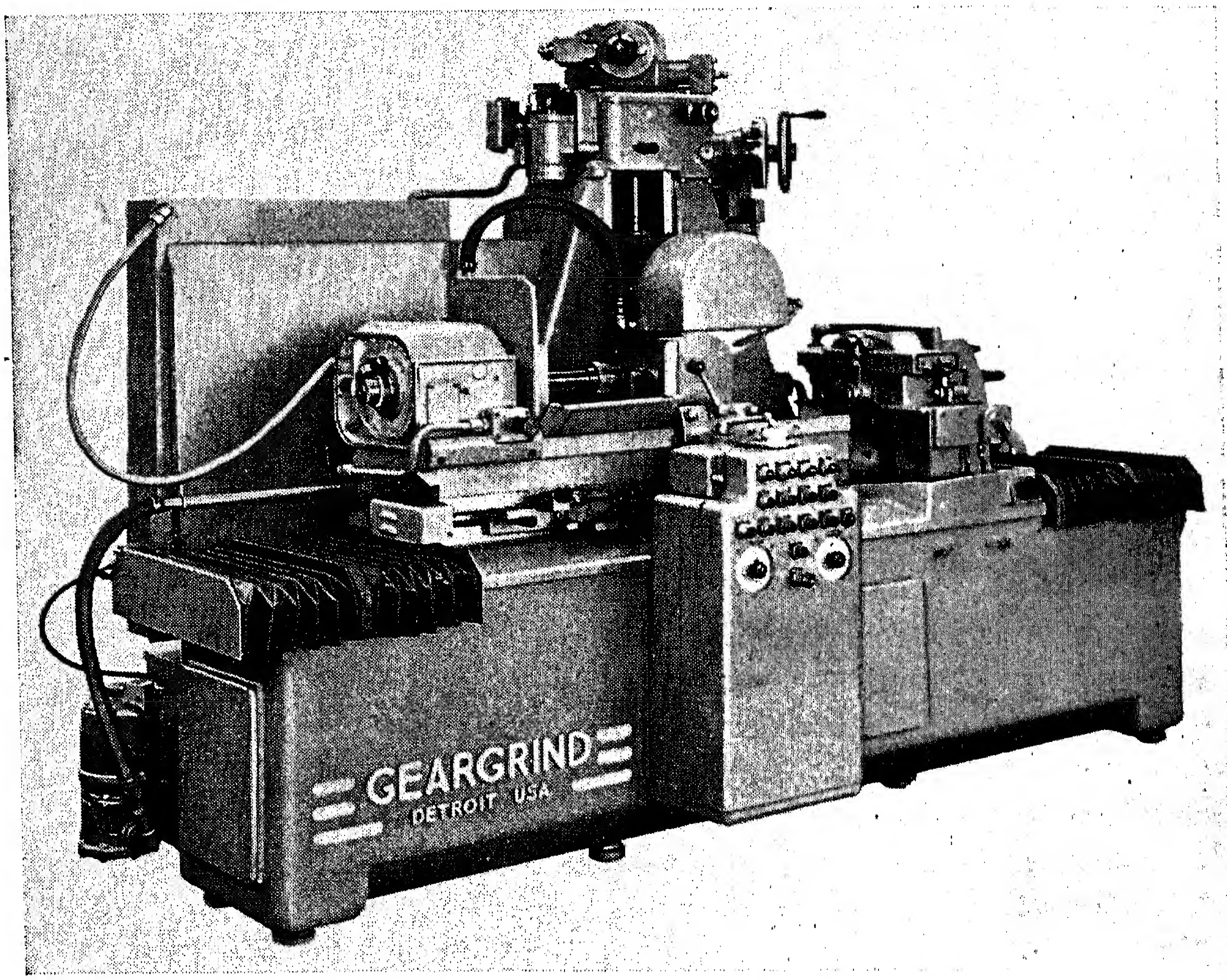


Fig. 20-17. A 10 in. by 24 in. formed wheel gear grinding machine. (Courtesy The Gear Grinding Machine Co.)

Gear tooth grinding. Hardened gear teeth are ground by two methods, forming and generating. The formed wheel on the gear grinder of Fig. 20-17 is trued by three diamonds on a truing device on the right end of the work carriage. The diamonds are guided through a pantograph mechanism by templates six times the gear tooth size. A workpiece between centers or on a stub arbor and connected to an index head is reciprocated under the grinding wheel, which is fed down at each stroke until desired size is reached. Then the workpiece is cleared from the wheel and indexed for the next tooth space. After a preset number of teeth has been reground in

this way, the wheel is trued. All these functions are performed automatically. Usual procedure is to rough grind around a gear, true, and then finish grind. The grinding wheel is set to depth manually for finish grinding, and the work is reciprocated and indexed automatically until all teeth are finished. Formed wheel gear grinders are also made for internal gears.

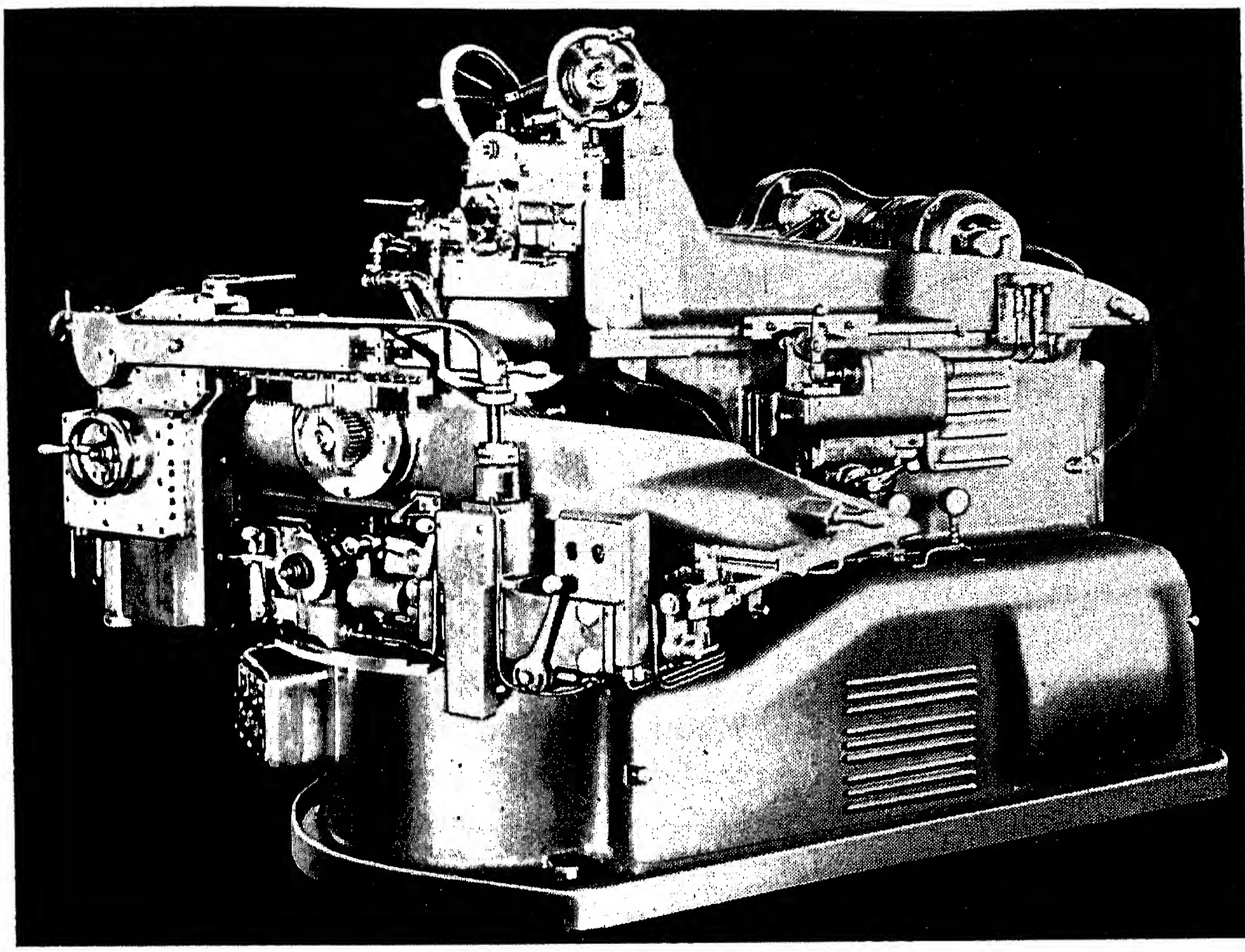


Fig. 20-18. A 10 in. hydraulic gear grinder for spur and helical gears. (Pratt and Whitney Photo from Pratt and Whitney Division, Niles-Bement-Pond Co., West Hartford, Conn.)

The wheel on the generating-type gear grinder of Fig. 20-18 is trued to represent a tooth of a basic rack. The workpiece is mounted on an arbor or between centers and is rolled past the grinding wheel which is reciprocated to cover the full gear width. The rolling action is governed by a master gear on the work spindle engaged with a rack on the front of the machine. The workpiece is indexed automatically when it has rolled out of engagement with the grinding wheel. A spur gear is ground with its axis in line with the

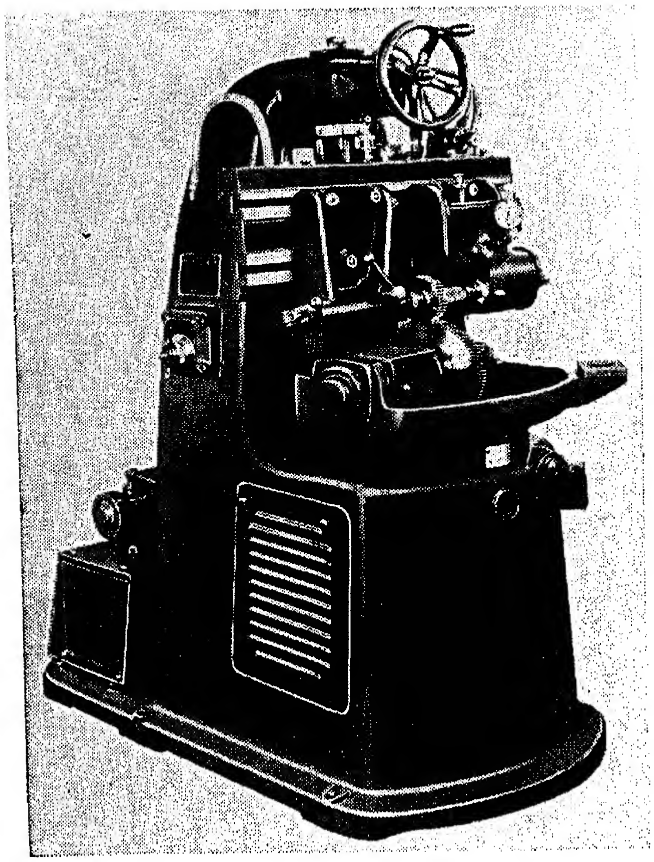


Fig. 20-19. A gear lapping machine. (Courtesy National Broach and Machine Co.)

wheel movement, and a helical gear is swiveled. Another model of generating gear grinder operates on the same principle but has two wheels that are not reciprocated but are large enough to cover an entire tooth working surface on gears not more than about 1¼ in. wide. It is faster for work of that kind than the one wheel grinder.

Gear tooth lapping. Gears that are finish cut or shaved and then heat treated are commonly lapped afterwards to remove scale and correct small errors of distortion. A gear may be lapped by running it with one or more cast iron toothed laps under a flow of fine abrasive in oil. On the machine of Fig. 20-19, the lap meshes with and drives the gear above it. Their axes are crossed, and the workpiece is reciprocated axially to increase sliding between the tooth surfaces. The work is turned first in one direction and then in the other, to lap both sides of the teeth.

Gears may be finished by cramp lapping or power tailstock lapping. *Cramp lapping* is the fastest way to correct eccentricity and index errors. The workpiece is forced against the revolving lap but otherwise runs free. In *power tailstock lapping*, the gear and lap are run at a fixed center distance, and a hydraulic brake retards the gear rotation. This may be arranged to lap the gear teeth more on the driving side than the other.

On one type of lapping machine, the workpiece is run in mesh with three equally spaced laps, all at crossed axes. The method is fast and breaks up tooth spacing errors.

A gear lapping operation usually takes several minutes. The laps are subject to wear, but on average work a lap can finish from one to three thousand gears. Good quality hard spur or helical gears can be manufactured at the lowest cost by shaving, heat treating, and

lapping. The highest quality gears are produced by grinding after heat treatment, but the cost is higher.

Hardened straight and spiral bevel and hypoid gears are lapped by running them with mating pinions under load to duplicate operating conditions. Abrasive in oil is poured on them. The gears are oscillated and rotated in one direction and then the other. Adjustments are made to obtain a desirable bearing on the teeth.

Gear Inspection

The inspection and measurement of gears involve some techniques not common for other products. Gears are tested for:

1. the accuracy of linear dimensions such as outside and root diameters and tooth thickness and depth
2. tooth profile
3. positions of the teeth as reflected by tooth spacing, runout, radial position, backlash, and helix angle or lead
4. the bearing and finish of the tooth faces
5. noise

Checking the sizes of gears and gear teeth. A *gear tooth vernier caliper* measures the thickness of a gear tooth at the pitch line, as is done in Fig. 20-20. A blade between two jaws is set up a distance equal to the tooth addendum by means of the vertical vernier scale and is placed on the top of the tooth. The jaws are brought together to touch the sides of the tooth, and the thickness is measured by the horizontal vernier scale.

A *gear tooth comparator* has two jaws that are set a proper distance apart by means of a master tooth. The jaws are then placed against the sides of a tooth to be checked, and an indicator registers the addendum height of the tooth.

Ground rolls of accurate diameter to make theoretical contact at the pitch line may be placed in opposite tooth spaces of a gear. The size of the gear is checked by measuring the distance across the rolls with a micrometer. Formulas and tables of proper roll sizes and measurements are given in handbooks.

Checking gear tooth profile. An optical comparator like the one in Fig. 3-35 offers one way of checking the profiles and positions of

gear teeth from enlarged charts. Fixtures are commonly used to position the gears.

The machine of Fig. 20-21 for measuring gear tooth involute profiles has a vertical spindle carrying the gear above a disk of the same diameter as the base circle of the involute to be checked. The disk bears against a bar on a slide on the front of the machine. The slide carries an indicator and tracer finger that touches a gear tooth at a point directly above the edge of the bar which corresponds to a line of action. As the slide is moved, the disk and gear roll together, and the indicator registers deviations in the involute profile of the tooth. Some machines are equipped with an electrical recorder to trace the involute form on a chart.

Checking the positions of gear teeth. A typical device to check tooth spacing carries the gear freely between centers. A tapered block is brought in on a

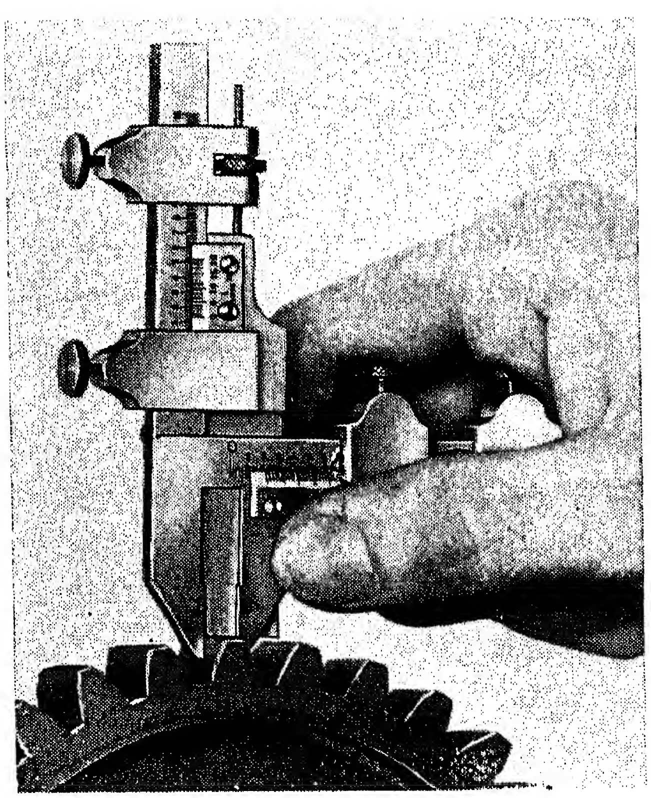


Fig. 20-20. An application of a gear tooth vernier caliper. (Courtesy Brown and Sharpe Mfg. Co.)

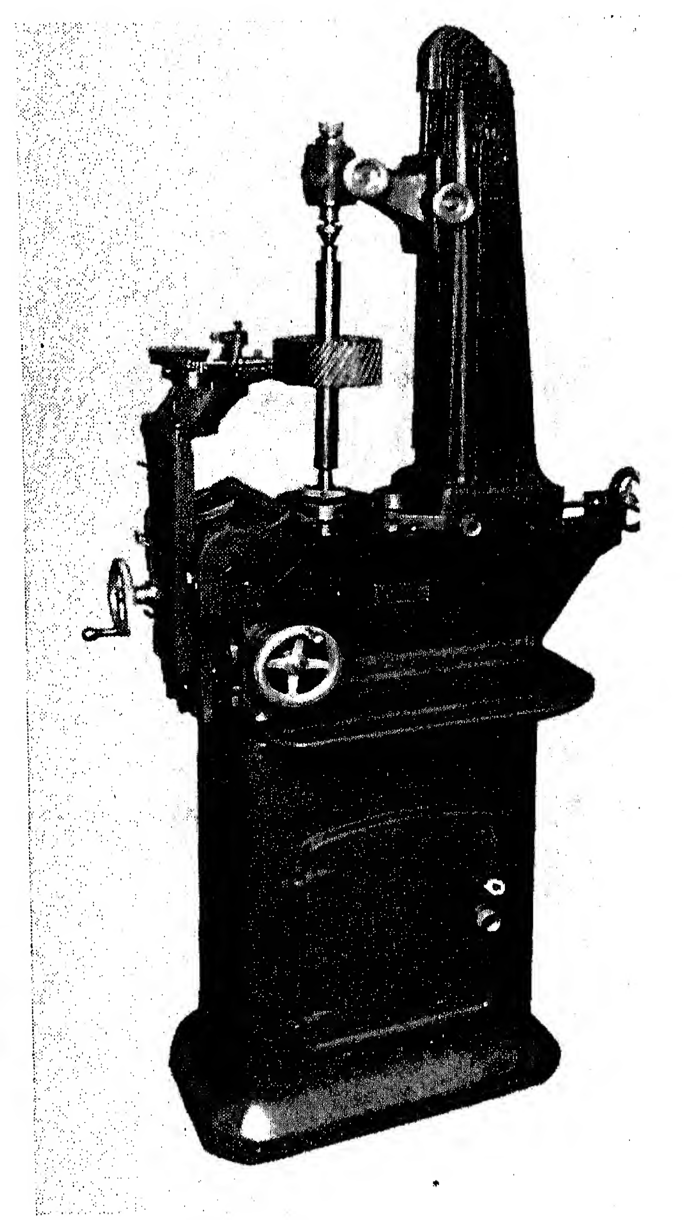


Fig. 20-21. An involute profile measuring machine. (Courtesy Illinois Tool Works Co.)

slide to locate a tooth space. An indicating finger previously set to a master makes contact with the face of the next tooth. Error in tooth spacing displaces the finger and is indicated on a dial.

The lead and helix angle of a helical gear may be compared with a master cam through an indicating mechanism on one type of machine.

To check backlash, a work gear and master gear may be mounted

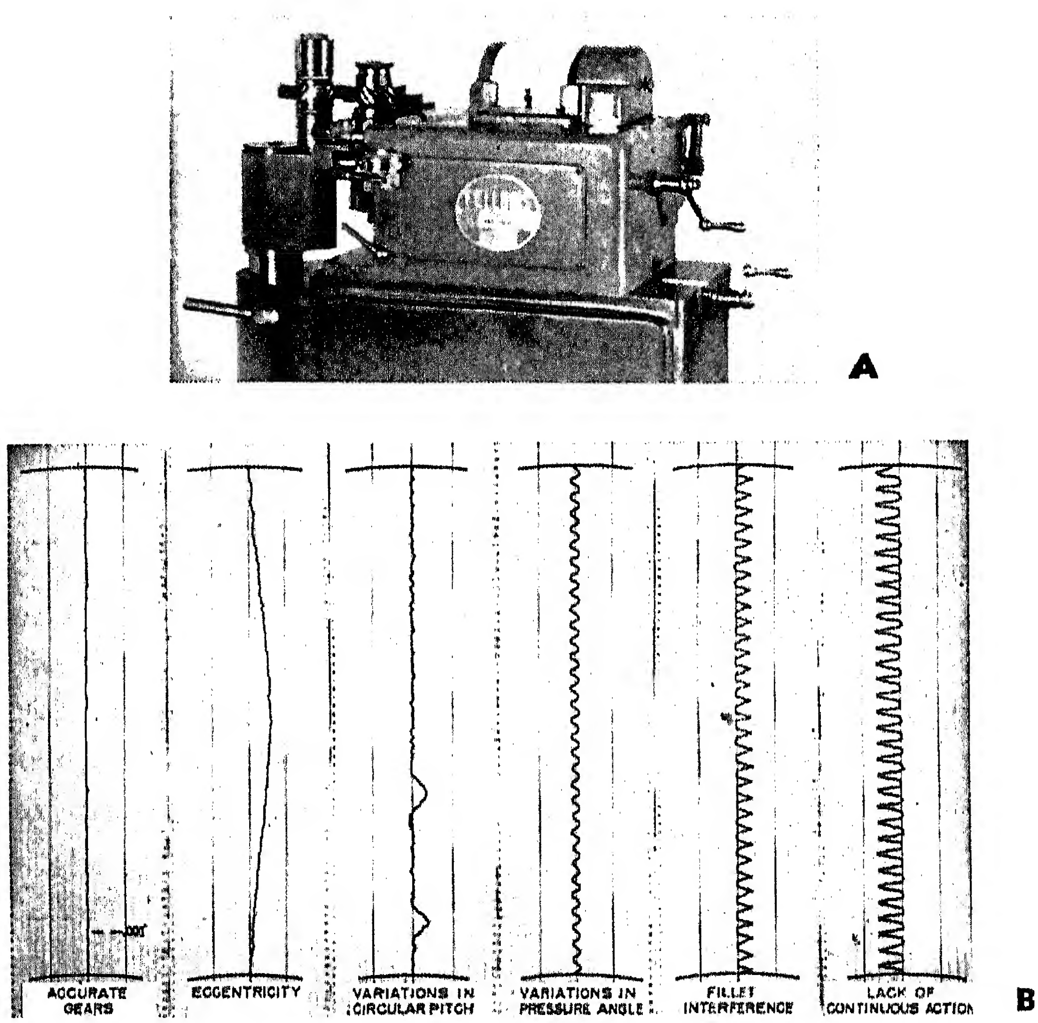


Fig. 20-22. A. A Red Liner gear checker. B. Examples of Red Liner chart indications. (Courtesy The Fellows Gear Shaper Co.)

on shafts at fixed center distances. The master gear is held still, and the slight movement or backlash of the engaged gear teeth is measured by an indicator.

Composite errors in tooth spacing, thickness, profile, runout, interference, and eccentricity are reflected in changes either in velocity or center distance when gears are run together. A common way

of checking these errors is to run a gear with a master. One gear shaft is fixed and the other can move, but the two are drawn together by a spring. The *Red Liner Gear Checker* of Fig. 20-22 A operates on this principle. Movement of the master gear shaft as the gears are run together is magnified and recorded on a chart. Each cause of error can be identified from the features of the curve traced on the chart, as depicted in Fig. 20-22 B.

Checking gear tooth bearing and surface finish. Bevel and hypoid gears commonly are inspected by running them together or with masters to determine where the teeth bear. Localized tooth bearing near the center of each face is desirable to avoid concentrating loads at the ends of teeth. The teeth are painted with marking compound to show up the areas of contact. Machines and fixtures are available for running the gears together under proper conditions.

The surfaces of finished gear teeth may be checked with a profilometer.

Checking for noise. Faulty gears are noisy. Machines are available for running gears together to test them for noisiness. Adjustments are provided for various types and sizes of gears. Power is applied to one gear, and a brake loads the other. A horn or other suitable means serves to collect the sound and direct it to the inspector's ear. Characteristic sounds are designated by such descriptive names as squeal, whine, growl, knocks, nicks, and marbles, and each indicates certain kinds of gear errors to an experienced inspector.

Questions

1. What curve is found on most gear teeth? What are its advantages?
2. Name the important elements of a gear tooth and state how they are determined for full depth and stub teeth.
3. Describe the common gear tooth systems.
4. Define spur gear, helical gear, herringbone gear, worm, worm gear, bevel gear, crown gear, and hypoid gear.
5. What are the four general methods for making gears?
6. What are the three methods of cutting gear teeth?
7. Describe how a spur gear is cut on a milling machine. What are the advantages and limitations of this method?
8. How may gears be form cut in production?

9. Describe a hobbing machine and the hob used on it?
10. Describe a vertical gear shaper and the pinion-type cutter used on it?
11. Compare gear hobbing and shaping with each other and with other methods of gear cutting.
12. Describe the actions of the two basic machines of universal type for cutting bevel gears.
13. Why are gear finishing methods used after gear cutting? What are the conventional methods of gear finishing?
14. In what two ways may gears be shaved?
15. How and why are gear teeth ground?
16. When and how are gear teeth lapped?
17. For what errors are gears tested? Describe the common methods of checking gear teeth.

Problems

1. A soft steel spur gear with 40 teeth, 3.500 in. outside diameter, and 1 in. width is to be cut on a milling machine. What should be:
   (a) the diametral pitch, addendum, tooth thickness, and whole depth?
   (b) the specifications of the cutter?
   (c) the number of turns of the dividing head crank to index each tooth?
   (d) the cutting time?
2. A 30 teeth spur gear is to be hobbed with a 4 in. diameter hob turning at 90 sfpm. What is the work speed in rpm if the hob has a single thread? A double thread?

References

American Standards Association, 70 E. 45th St., New York 17, bulletins:
B6.1–1932 Spur Gear Tooth Form.
B6.2–1933 Gear Materials and Blanks.
B6.3–1949 Backlash for General Purpose Spur Gearing.
B6.5–1949 Letter Symbols for Gear Engineering.
B6.10–1950 Gear Nomenclature.
B6.6–1946 Gear Tolerances and Inspection.
B6.7–1950 20 Degree Involute Fine-Pitch System for Spur and Helical Gears.
B6.8–1950 Fine-Pitch Straight Bevel Gears.
B6.9–1950 Design for Fine-Pitch Worm Gearing.

Buckingham, Earle, *Spur Gears — Design, Operation, Production.* New York: McGraw-Hill Book Co., 1928.

Colvin, F. H. and Stanley, F. A., *Gear Cutting Practice.* New York: McGraw-Hill Book Co., 1943.

Davenport, G., *The Hobbing Method of Cutting Gear Teeth.* New York: McGraw-Hill Book Co., 1940.

———, *The Face Gear.* Springfield, Vt.: The Fellows Gear Shaper Co.

———, *The Involute Gear.* Springfield, Vt.: The Fellows Gear Shaper Co.

———, *Modern Methods of Gear Manufacture.* Detroit: National Broach and Machine Co., 1950.

Trautschold, R., *Standard Gear Book.* New York: McGraw-Hill Book Co., 1935.

Manufacturers' Catalogs:

Barber-Colman Co., Rockford, Ill.
Boston Gear Works, Boston, Mass.
Brown and Sharpe Mfg. Co., Providence, R.I.
Cleveland Hobbing Machine Co., Cleveland, Ohio.
Farrel Birmingham Co., Ansonia, Conn.
Fellows Gear Shaper Co., The, Springfield, Vt.
Foote Bros. Gear and Machine Co., Chicago, Ill.
Gleason Works, Rochester, N.Y.
Gould and Eberhardt, Inc., Irvington, N.J.
Gear Grinding Machine Co., The, Detroit, Mich.
Illinois Tool Works Co., Chicago, Ill.
Michigan Tool Co., Detroit, Mich.
National Broach and Machine Co., Detroit, Mich.
Pratt and Whitney Division, Niles-Bement-Pond Co., W. Hartford, Conn.

Chapter 21

TURRET LATHES AND AUTOMATICS

OUT OF THE SIMPLE LATHE, MACHINES HAVE EVOLVED with refinements that make them efficient for producing duplicate parts in quantities. A basic feature of such production machines is that they can be arranged to repeat certain functions. One of these, the automatic lathe, has been described and illustrated in Fig. 5-8. Evolution along other lines has led to turret lathes and automatic screw and chucking machines.

Turret lathes are lathes with multiple toolholders that enable all the tools for a particular operation to be preset and presented to the work repeatedly as needed. An operator must give full attention to a turret lathe, but he can produce faster than on an engine lathe where more than one or two tools are involved. Turret lathes are economical for producing parts in moderate quantities.

Automatic turret lathes, screw machines, and chucking machines not only have means to position a variety of tools repeatedly but do so without constant attention from an operator.

Generally, a machine tool that is faster than another for a job is more complex, takes more time for setup, and costs more. Such a machine is justified only if enough pieces are required to enable it to save in direct labor an amount larger than the extra setup. maintenance, and capital charges incurred.

Turret Lathes

A turret lathe is so named because it has a hexagonal turret in place of a tailstock and often a square turret on the cross slide in-

stead of a single toolpost. Also a toolpost may be mounted on the rear of the cross slide. Thus tools can be mounted in a number of stations and positioned readily for cutting as needed. Work may be done on bar stock or on individual pieces. Sometimes the name of *screw machine* or *hand screw machine* is given to a turret lathe equipped for bar stock, particularly in the smaller sizes.

The name turret lathe alone ordinarily applies to a horizontal machine. Vertical turret lathes usually are so designated and are

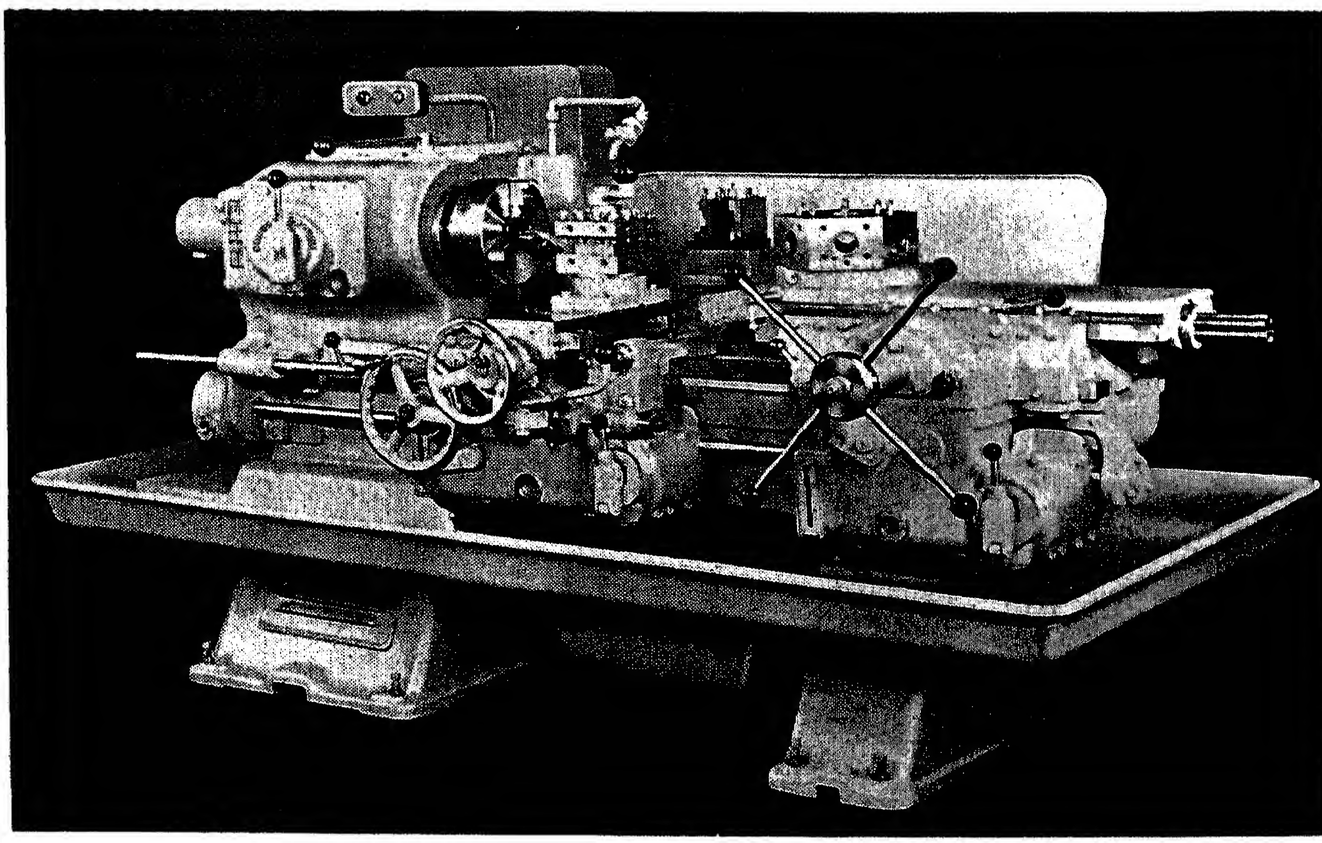


Fig. 21-1. A No. 5 ram-type turret lathe. (Courtesy Jones and Lamson Machine Co.)

described in Chapter 14. The two main types of horizontal turret lathes are the *ram type* and the *saddle type.*

Ram-type turret lathes. A typical ram-type turret lathe without tooling is illustrated in Fig. 21-1. The hexagonal turret is carried on a ram that slides longitudinally on a saddle positioned and clamped on the ways of the bed. Each of the six faces of the turret has a straight hole for locating tools and holders. Clamps extending above the top of the turret fasten the tools in the holes. Toolholders with flanges are bolted to the faces. The tools on the face toward the

headstock are fed to the work when the ram is moved to the left on the saddle. When the ram is moved to the right, the turret indexes clockwise to the next station and is locked in position. The ram is lighter and can be moved more quickly than a saddle but lacks some rigidity. Because of convenience and speed, the ram-type construction is favored for small and medium-size turret lathes where the ram does not have to overhang too far.

The headstock on the left end of the bed is like that on an engine lathe, only heavier. Turret lathes often are driven by larger motors than engine lathes of comparable sizes to provide power for heavy production cuts. A 15 hp motor is recommended for the turret lathe of Fig. 21-1. Two types of headstocks are the *electric head* and the *all geared head*. In an electric head a multiple speed electric motor drives the spindle directly at relatively high speeds. The turret lathe of Fig. 21-1 has an all geared head with a speed *preselector* dial and lever. This is a device that enables the operator to select the speed for the next cut while waiting for a cut to be finished. As soon as one cut is finished, the operator pushes the starting lever to change the speed, and the next cut can start at once. Most modern machines offer 12 spindle speeds.

The cross slide moves on a carriage on the ways between the headstock and ram. The type in Fig. 21-1 is commonly found on ram-type turret lathes and is called the *reach-over* or *bridge type*. Its carriage rides on both the front and rear ways on top of the bed and may also be supported by a lower way on the front of the bed. A quick hand-indexed four-station turret is commonly mounted on the front of the cross slide, and a holder for one or more tools often is mounted on the rear. Heavy tools with shanks of square or rectangular sections are clamped in the side slots by the binding screws in the top of the turret. The tools may be adjusted for height by rockers or shims. The tools are positioned and fed to the work by the movement of the carriage along the bed and the cross movement of the slide. A *plain* cross slide is entirely hand operated, but the *universal* kind is power fed also and is more common on turret lathes, as typified by Fig. 21-1. The cross slide can be positioned accurately by means of a cross screw and micrometer dial.

The power feed to the carriage and cross slide of the turret lathe of Fig. 21-1 is reversible in direction. Nine feeds are available, and selection is made through the lever and dial at the lower right-hand

corner of the carriage apron. Above the feed selector dial on the apron is the lever that is raised to engage the cross-feed. Above that lever is an indexing four-position stop roll with a screw for each position. Each of the screws can be adjusted for a cut. Then one screw at a time is positioned so that it is hit by a trip dog on the side of the cross slide. That causes the feed lever to drop out of engagement and stops the feed as desired for the particular cut.

A stop bar is held by a bracket on the front of the bed below the headstock. This bar can be clamped in any convenient position and has an extension that can be set and locked in any one of a number of positions. It butts against adjustable screws in an indexing six-position roll stop on the left side of the carriage apron. This trips the carriage feed and provides a means to position the tools along the work. The carriage can be locked to the bed after being positioned.

The ram is moved by hand by means of the capstan lever on the saddle. Nine power feed rates also are available and are selected from the dial by the lever at the lower right-hand corner of the saddle apron. A stop roll with long adjusting screws is carried on the rear end of the ram. This roll indexes in unison with the turret so that the bottom screw always corresponds to the face of the turret turned toward the work. That screw trips the power feed and stops the ram movement in the direction of the work.

Most turret lathes have means for flooding the work and cutting tools with cutting fluid. A cutting fluid line with several joints to position the spout anywhere in the working area is seen over the front of the headstock in Fig. 21-1. In addition, turret lathes often have cutting fluid piped to the center of the turret and supplied to the tools in cutting position.

The size of a turret lathe is designated by a number that indicates the bar and swing capacity, but different makes of machines with the same size number may vary somewhat in capacity. The No. 5 ram-type turret lathe of Fig. 21-1 has a maximum round bar capacity of 2½ in., is equipped with a 12 in. three jaw universal scroll chuck, and swings a 13 in. diameter over the cross slide and an 18½ in. diameter over the carriage ways. No. 5 turret lathes made by others may have slightly larger or smaller dimensions. Some manufacturers are resorting to designating sizes by double numbers, such as 5-3 which specifies a 3 in. diameter maximum bar capacity. Turret

lathes of different makes but comparable size differ with regard to ease of operation, rigidity, and accuracy.

Saddle-type turret lathes. The hexagonal turret of a saddle-type turret lathe is carried directly on a saddle that slides lengthwise on the bed as depicted in Fig. 21-2. This construction is favored for large turret lathes because it provides good support and a maximum cutting range for the turret tools. The saddle is moved toward the

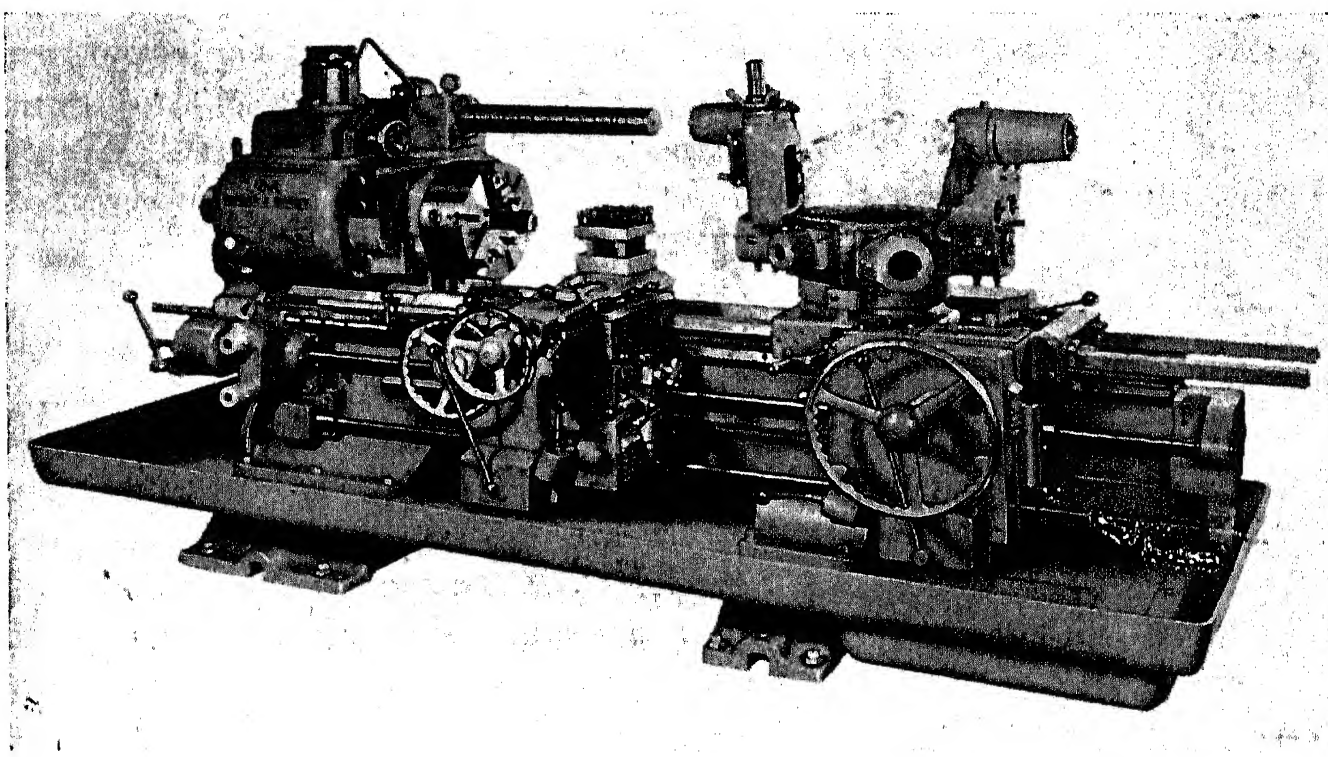


Fig. 21-2. A No. 2-A saddle-type turret lathe. (Courtesy The Warner and Swasey Co.)

headstock by hand or power to feed the tools to the work, and the turret is indexed when the saddle is withdrawn. The stop mechanism to trip the feed and stop the saddle as desired for each turret position is located between the bed ways and is attached to the saddle unit. One make of turret lathe has auxiliary stops that can be used for any turret position in addition to the regular stops.

The turret is fixed in the center of the saddle on some machines. On others, it is mounted on a cross slide that may be moved by hand or power on the saddle and offset an accurate distance by a cross screw and micrometer dial. Overhang of the tools is kept down by setting the turret off center to machine large diameters or

faces. The cross-sliding turret is also helpful for taper or contour boring and turning.

Some saddle-type turret lathes have the reach-over type of cross slide, but most are equipped with the *side-hung* type shown in Fig. 21-2. The carriage slides on ways at the top and bottom on the front of the bed and does not extend to the rear way. That allows large pieces to be swung without interference from the cross slide, but the rear tool station on the cross slide is lost. The side-hung type of cross slide is provided with hand and power feeds and stops. In addition, the large machines commonly have power rapid traverse feed to the carriage, cross slide, and saddle. A compound slide on the cross slide with hand and power feed is available on some machines.

Turret Lathe Tools and Attachments

Standard work-holding devices commonly used on turret lathes include collets and collet chucks for bar stock and hand and power chucks for individual pieces. Spring-type collets are used for bar stock up to about 2½ in. diameter and for second operation work because of their accuracy. Parallel closing-type collet chucks with interchangeable jaws of various sizes are desirable for large diameters because they make full contact and are adjustable to the variations in large bars. The bar stock may be fed through without stopping the spindle, usually by bar feeding devices. The machine of Fig. 21-4 is equipped with a bar feeding device. Chucks include those with 2, 3, or 4 jaws, sometimes independent, but usually universal like those found on engine lathes.

Special fixtures often are used instead of standard devices for irregularly shaped pieces, added rigidity, and moderately large quantities. They may be attached to face plates.

Essentially the same kinds of cutting tools are used on turret lathes as on engine lathes. Tools for holes include twist drills, boring bars and cutters, reamers, and taps. Large single point tools with integral shanks are mounted on the cross slide for heavy turning and facing. Cross-slide form tools are popular because they provide fast means for producing finished shapes. They may be forged tools ground to shape, straight tools held by a dovetail, or circular forming tools.

Convenient devices are made specifically for holding and adjusting cutting tools singly and in groups on the hexagonal turrets. This equipment is available commercially in many standard forms and sizes and is fully described in manufacturers' catalogs. Similar tools are used on automatic screw and chucking machines. A few typical kinds are presented here. Some equipment is intended for bar work, other for chucking work, but much is fitted for both kinds of service.

Bar stock generally is supported by the tailstock center on an engine lathe, but that usually is not feasible on a turrent lathe, especially when cuts are taken from the hexagonal turret. *Box tools* support overhanging bars that are being turned, faced, chamfered, or centered from the hexagonal turret. Rollers or a crotch bear against and back up the work surface opposite the cutting tool. The *single cutter turner* of Fig. 21-3 A is an example of a box tool. Other models are arranged to carry several turning bits, a facing bit, or a center drill.

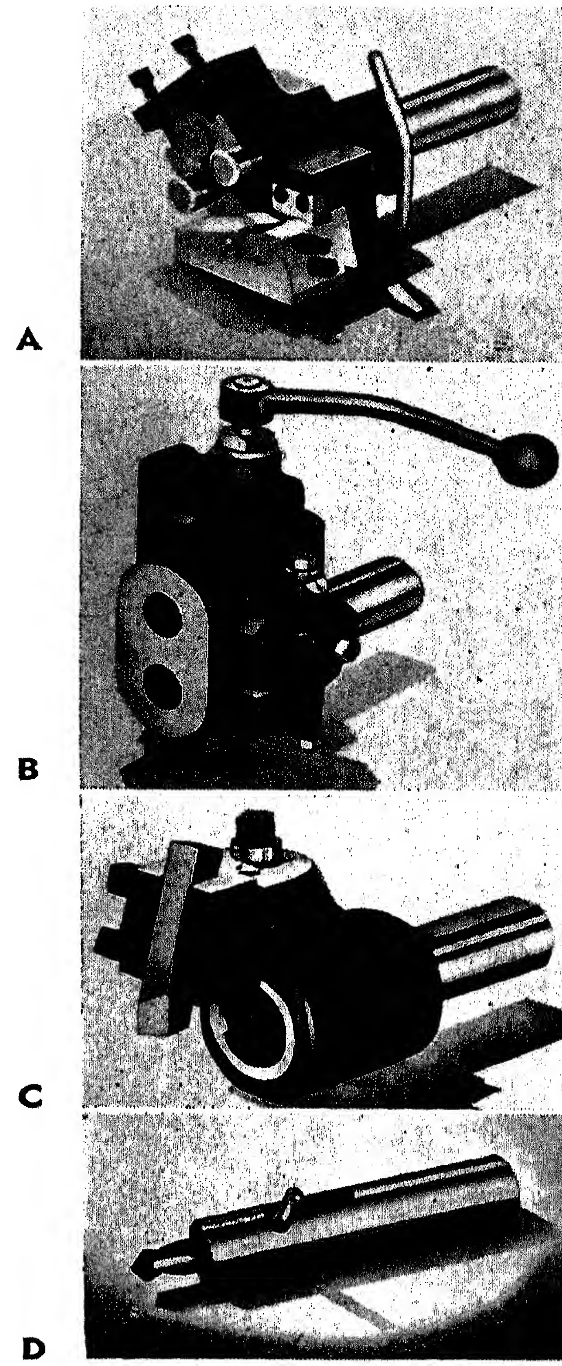


Fig. 21-3. Examples of tooling equipment for bar work. A. Single cutter turner. B. Quick acting slide tool. C. Adjustable knee tool. D. Combination stock stop and starting drill. (Courtesy The Warner and Swasey Co.)

The *quick acting slide tool* of Fig. 21-3 B is designed to hold round shank single point tools and boring bars for fast recessing and facing cuts on both bar and chuck work. The slide that carries the cutting tools moves ½ in. with a quarter turn of the handle. The *adjustable knee tool* of Fig. 21-3 C can be set up quickly for turning combined with drilling, boring, or centering on short bars. The *combination stock stop and starting drill* is a two-purpose tool that saves one turret face and one index. A similar tool has a center in-

stead of a drill. *Flanged toolholders* are made in two lengths as shown on the side of the turret in Fig. 21-2 and in several styles. They serve as adapters and extension toolholders for drills, boring bars, and reamers. Other toolholders include bushings, sleeves, sockets, floating reamer and tap holders, and drill chucks. A typical setup of bar equipment is illustrated in Fig. 21-4.

Except for long and large diameter threads, die heads are used for external threads and taps for internal threads on a turret lathe. When desired, a thread chasing attachment may be added and ap-

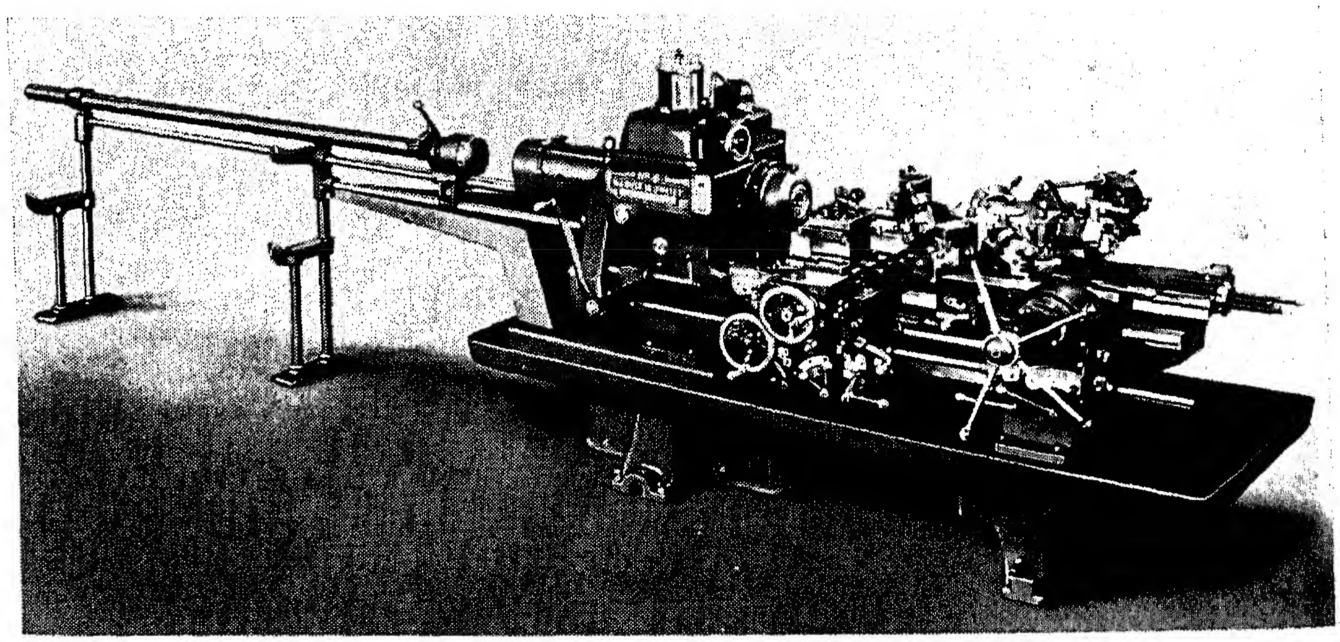


Fig. 21-4. A ram-type turret lathe set up for bar work. (Courtesy The Warner and Swasey Co.)

plied to the carriage for chasing threads with a cross-slide tool or to the ram slide or saddle for leading taps and die heads. The movement of the tool is controlled by a screw and nut, as on an engine lathe.

Chucking work often involves a greater range of diameters and more tool overhang than bar work. The chucking setup in Fig. 21-5 contains several characteristic tools. An *adjustable single turning head* is mounted on the turret face nearest the chuck. It carries a boring bar, a turning tool on an adjustable slide, and a *rotating overhead pilot bar*. A *multiple turning head* on the station farthest from the chuck carries a drill, an *adjustable angle cutter holder* and bit, and pilot bar. The pilot bars slide into a bushing on the headstock when the tools are in cutting position and add rigidity to the

tooling. A *stationary pilot bar,* like the one in Fig. 21-2, is attached to the headstock and slips into bushings in the turning heads. It can be heavy and strong because it does not add to the load on the turret. Another form of pilot is the *internal* or *center pilot* for guiding and reinforcing boring bars. It is a bushing in the work spindle that fits a pilot diameter on the ends of some boring bars.

The slide tool on the station nearest the operator's position in

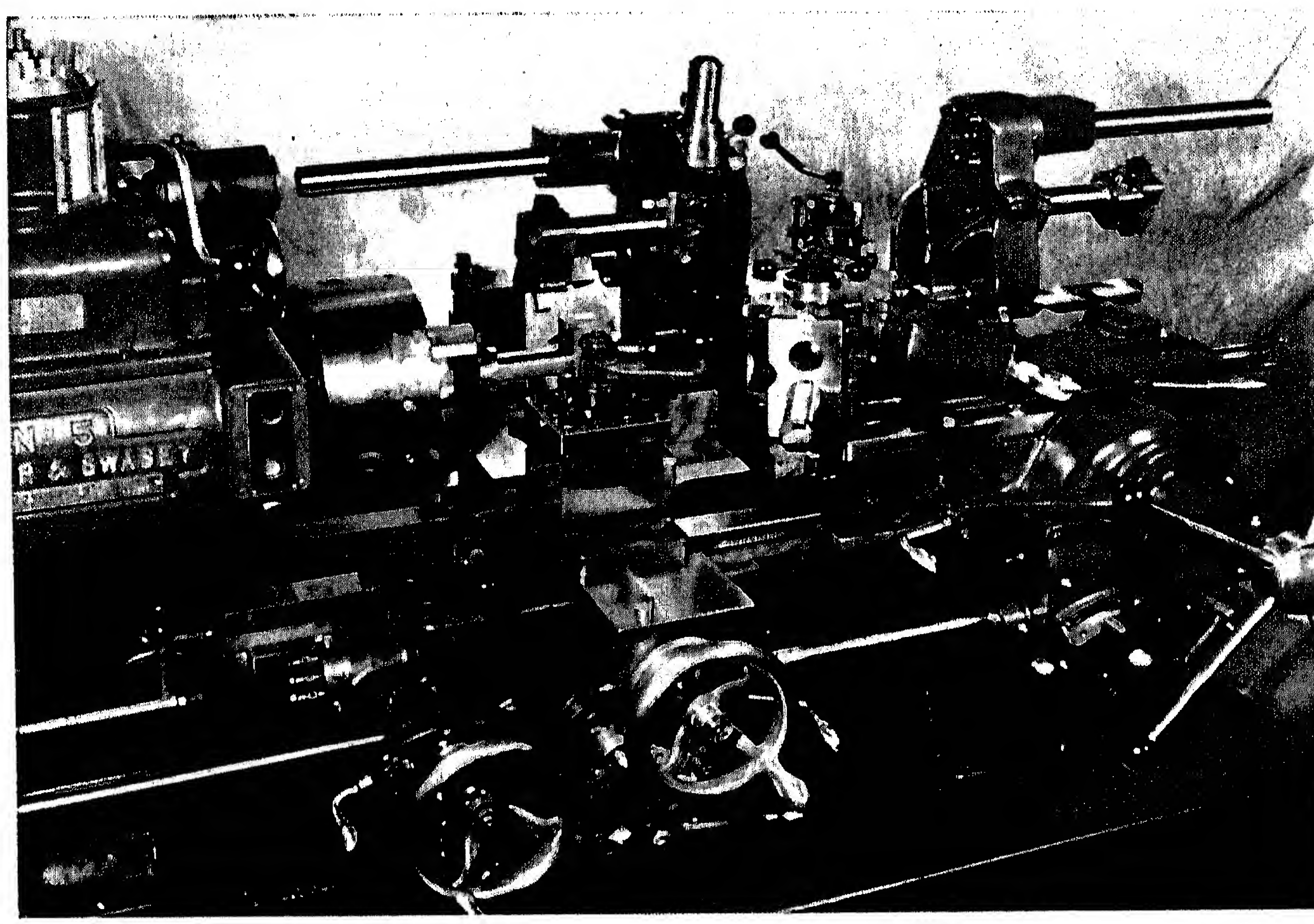


Fig. 21-5. A chucking tool set up. (Courtesy The Warner and Swasey Co.)

Fig. 21-5 provides a quick and accurate means of adjusting the boring bar. A quick acting slide tool is seen on the station farthest from the operator's position.

Turret Lathe Operations

Comparison of engine lathes, turret lathes, and automatics. A turret lathe does the same work as an engine lathe or automatic lathe, screw machine, or chucking machine but at a different rate.

The type of machine suitable for a job is logically the one that does the job at the lowest over-all cost. The governing factors are the nature of the workpiece and the number of pieces to be produced. The engine lathe is the normal choice to make one or a few pieces. If only one or two cuts are required on each piece, the engine lathe may be economical for a fairly large number of pieces because the time for changing and setting the tools is not large and the investment in the machine is small. On the other hand, if a number of tools must be applied to each workpiece, the turret lathe may prove best for a very few pieces.

The Warner and Swasey Co. manufactures both types of machines and has found from its experience that setup time on automatics is on the average three times as long as setup time on hand-operated turret lathes. Also on the average an automatic produces three times as many pieces per hour as a turret lathe. For the same labor rate and overhead on both machines, a rule of thumb is that a turret lathe is more economical for a job for which the quantity of pieces required is less than six times the number of pieces produced per hour on the turret lathe times the number of hours required to set up the turret lathe. Consideration should be given to the automatic for a larger quantity of pieces.

Selecting the machine and tools for a turret lathe operation. A turret lathe is seldom purchased for one job alone, and in most cases a production planner must make a selection from the turret lathes already available in his plant. The best machine is one that has ample but not too excessive capacity for the job to be done.

The choice of the work-holding device and the tooling for a turret lathe job depends largely upon whether the pieces are to be made from bar stock, forgings, or castings. Bar stock is convenient because it can be gripped behind the section from which a piece is cut, and all or most of the work can be done on the piece before it is cut off. Parts like collars, spacers, and gear blanks can be machined in groups, with operations such as drilling, boring, reaming, turning and cutting off done on all the pieces in each group in one gripping of the bar stock. The cost of a forging or casting is more than bar stock, but more material must be removed from bar stock for many parts, and they can be made faster from forgings or castings. As a rule, small quantities are made from bar stock and larger quantities from forgings or castings.

A turret lathe with a large spindle bore must be selected for large bar stock. Forgings and castings are held in chucks or fixtures, one at a time, and smaller turret lathes can be used. Bar stock calls for roller-rest-type turning tools; forgings, castings, and individual pieces need chucking-type tooling.

Planning for efficiency in turret lathe operations. Time is the major item of cost in a turret lathe operation. This includes the time to mount and adjust the tools, set up the machine, load and unload the workpieces, manipulate the machine in indexing and positioning the tools, and make the actual cuts. Definite provisions can be made to minimize each element of time to obtain maximum efficiency.

An efficient turret lathe operation is planned before it is run. An engineer makes a sketch of the turrets and tool stations and designs the setup for the operator. Tracing forms for sketching tool layouts quickly and clearly are found convenient in many plants. Representative job setup and analysis sheets are helpful guides for the planner.

Setup time may be kept to a minimum in changing from one job to another by using universal tooling and maintaining a permanent setup on a turret lathe. Large and heavy tools that perform basic functions on most operations are permanently mounted in their logical order on the turret. All these tools are not needed for every job, but then the turret may be back- or skip-indexed. In most cases the extra indexing time is less than would be taken to remove the tools and change them around on the turret. The lighter tools are rearranged in various combinations on the heavy toolholders for different jobs. Turret lathe manufacturers and users have made careful studies to find the best permanent setups for various classes of work and sizes and types of machines. Two recommended permanent setups for average jobs, one for bar work and the other for chucking work, are given in Fig. 21-6.

Internal cuts are almost always made by tools in the hexagonal turret of a turret lathe and are planned and set up first for an operation. Provision must be made for the proper order of internal cuts, for instance for drilling before boring and boring before reaming a hole. A study should be made to ascertain the minimum number of tools needed to obtain the required accuracy of internal diameters. The external cuts are arranged after the internal cuts to

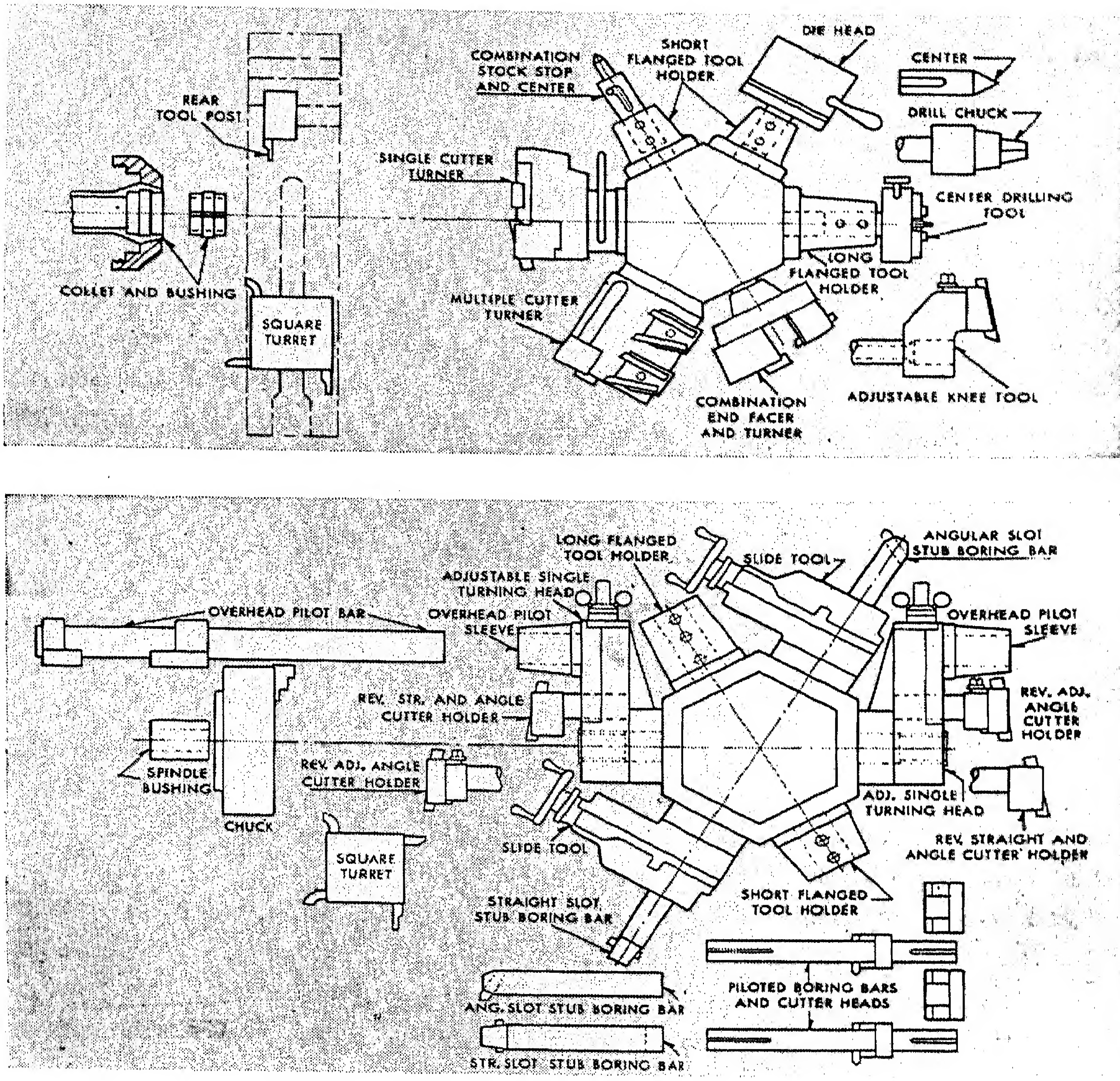


Fig. 21-6. Universal equipment for permanent set ups. (Courtesy The Warner and Swasey Co.)

take the best advantage of the available stations on the turrets. The preferred order of external cuts calls for turning, facing, and last of all chamfering, necking, grooving, and cutting off.

Work-handling time depends upon the proper selection of collets, chucks, and fixtures. For average work in small and moderate quantities, standard equipment is best with special jaws, arbors, and simple fixtures added as justified. Special fixtures can pay for themselves on single jobs where the parts are otherwise hard to hold and are made in fairly large quantities.

The selection of the proper size of machine is important for low machine handling time. More time is required to change setups and to index from station to station on a large machine than on a small one because of the heavier masses that must be moved.

Modern turret lathes are fully equipped with stops to position

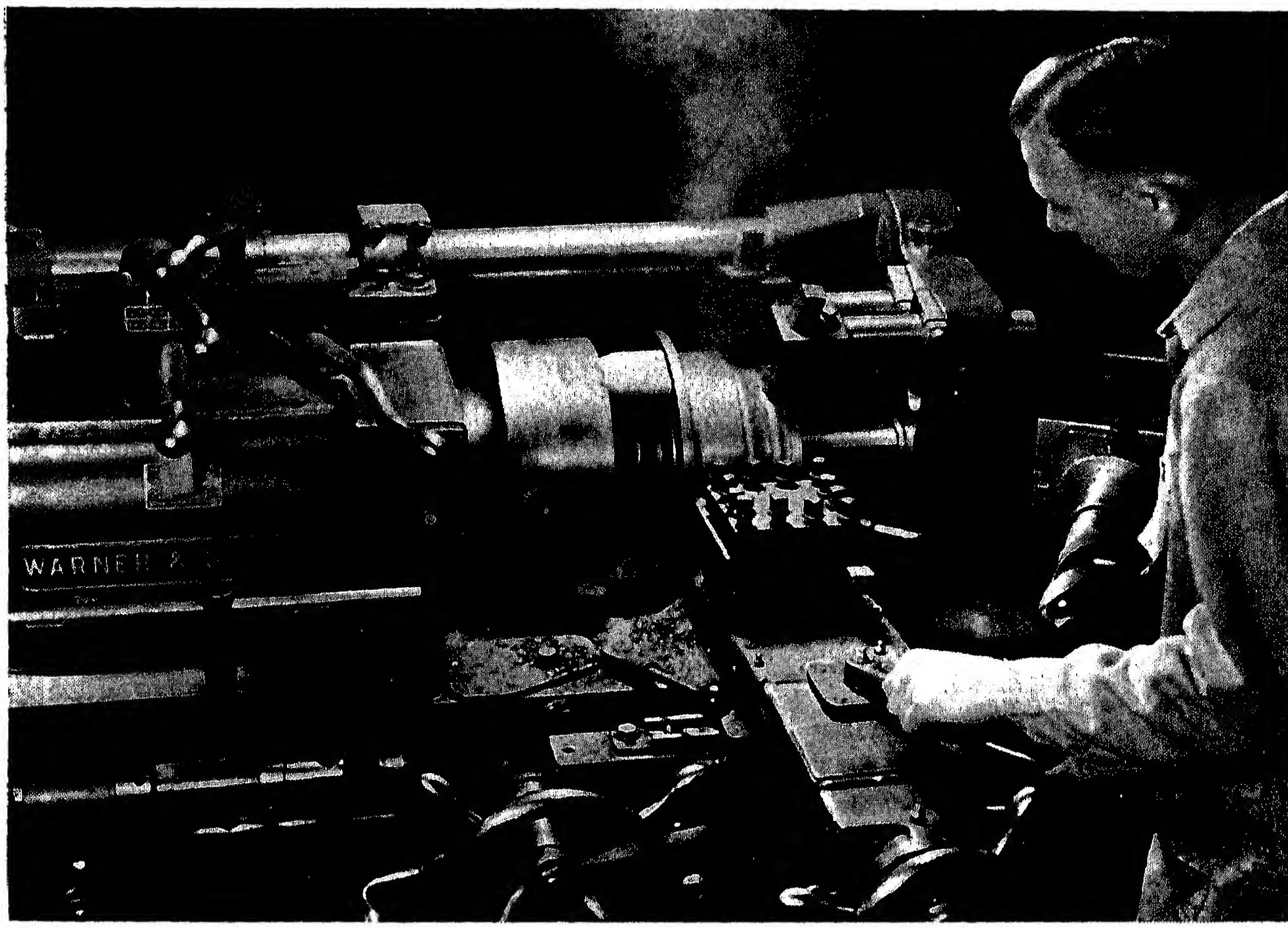


Fig. 21-7. Multiple internal and external cuts from the hexagonal turret combined with a cut taken by a tool in the square cross slide turret on a turret lathe. (Courtesy The Warner and Swasey Co.)

the cutters and trip the feeds. Full use should be made of these facilities to minimize machine handling time.

Machine handling time is reduced by taking combined and multiple cuts to save indexing. Cutting time is also reduced. A *combined cut* is one where tools in both the hexagonal turret and square cross-slide turret are made to cut at the same time. A *multiple cut* is one where two or more tools are applied at the same time from one turret station. An example of combined and multiple cuts is given in Fig. 21-7.

Full benefit can be had from combined and multiple cuts only if the tooling is substantial enough and the machine powerful enough for cuts to be taken together at about the same feed and speed feasible if they were taken one at a time. Full rigidity of tooling is obtained from rigid toolholders, minimum tool overhang, and the use of pilots.

Speeds, feeds, and cutting time for turret lathe operations. The same considerations govern speeds and feeds on turret lathes as have been discussed for lathes and other machine tools. For production purposes, turret lathes may be operated at higher speeds and heavier feeds than engine lathes. Recommendations for speeds and feeds for turret lathes are given in handbooks but serve only as starting guides, with adjustments to be made as the progress of each particular operation dictates.

The time to make each individual cut on a turret lathe is calculated in the same way as for cuts on an engine lathe.

Automatic Screw and Chucking Machines

A machine tool that moves the work and tools at the proper rates through a cycle to perform an operation on one piece is commonly called an *automatic*. Strictly speaking, the machine is a *semiautomatic* if an operator is required to unload and load the machine and start each cycle. Often an operator can do this for several machines in a group. A fully automatic machine tool is one that locates the tools in their correct working positions as needed, changes speeds and feeds for each pass, rejects the finished pieces, and presents new stock to the tools for each piece, all on its own accord. An attendant may have to load bars or fill a hopper with workpieces at intervals to keep a fully automatic machine supplied with material. Examples are given in this text of semiautomatic and fully automatic machines for turning, drilling, boring, milling, broaching, and grinding. Complete automation is found in some high production industries, where parts are passed mechanically from one automatic machine to another until completed.

Automatic machines for internal and external operations on bar stock are called *automatic screw machines*. They were originated for turning out screws but have been adopted for a multitude of

other products. Their counterparts for individual pieces are *automatic chucking machines.*

Automatic screw and chucking machines may be classified as single spindle or multiple spindle, with a number of variations in each class. Several typical kinds will be described.

Single spindle automatic screw machines. The popular single

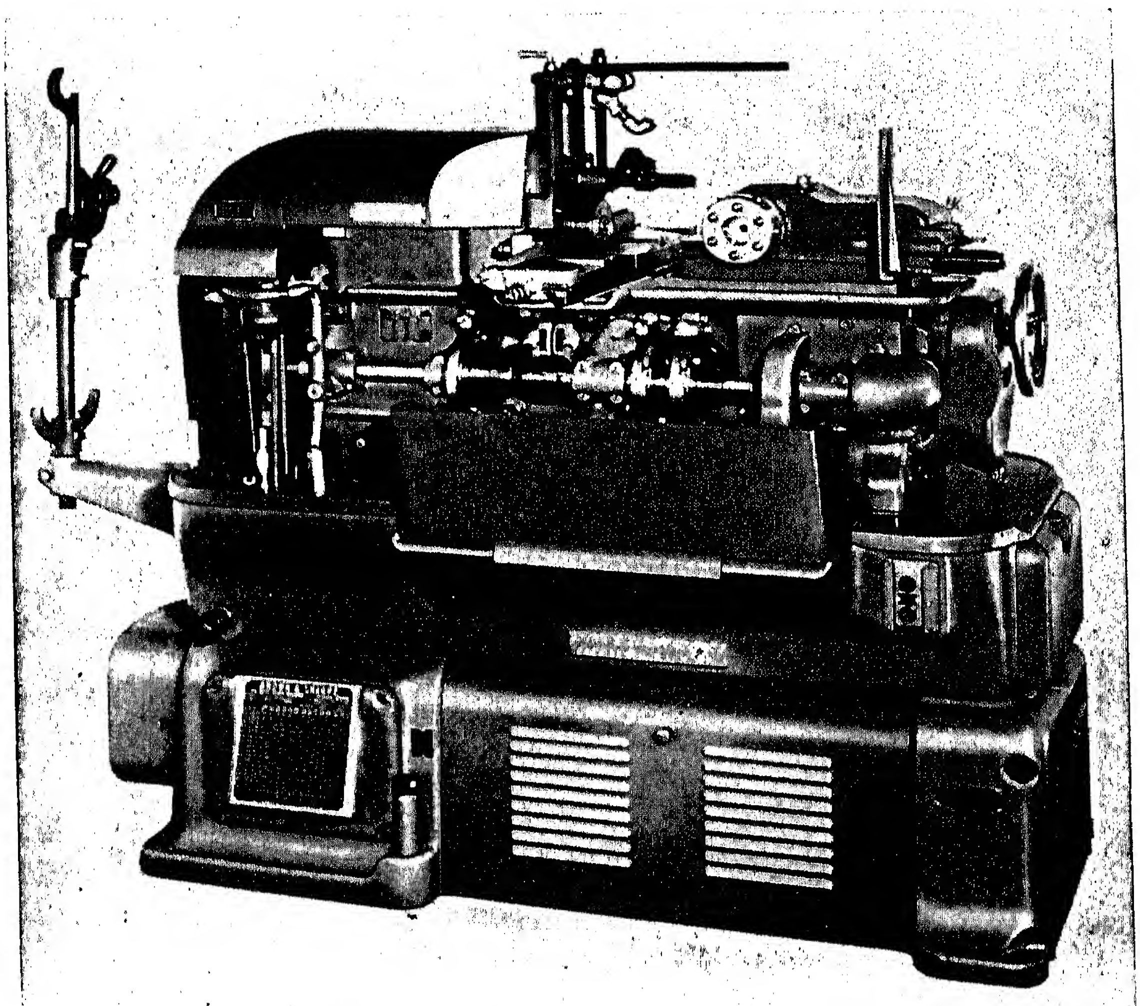


Fig. 21-8. A No. 2G single spindle automatic screw machine. (Courtesy Brown and Sharpe Mfg. Co.)

spindle automatic screw machine of Fig. 21-8 is made in four sizes to handle bar stock up to 1½ in. diameter and turn lengths up to 2 in. A 5 hp motor drives the one inch diameter capacity machine, which has a spindle speed range from 25 to 3025 rpm. Any one of 16 high forward speeds may be selected for a setup by means of change gears. For the same setup, any one of 12 low speeds, either forward or reverse, may be chosen. The power is transmitted by a chain drive to two loose pulleys on the spindle. Either pulley may be

connected to the spindle by a friction clutch to rotate the spindle as desired during the course of the operation at either the high or low speed selected.

Bar stock is fed through the revolving spindle of the machine of Fig. 21-8 by feed fingers for each cycle of an operation, is gripped in a collet, and is located by butting against a swing stop or turret stop.

Feed shafts run along the front, rear, and right end of the bed of the machine of Fig. 21-8. They carry clutches, dog carriers, and cams that actuate the machine movements. The shafts revolve at constant speeds regardless of the speed of the spindle and may be stopped at any time to stop the action of the tools. Dogs on the carriers on the front feed shaft engage clutches on the rear feed shaft at preset intervals. Power is applied through the clutches to change or reverse the spindle speed, feed the stock, and index the turret. Each of these functions takes only a fraction of a second, always at the same fast rate.

The turret has six stations, indexes around a horizontal axis, and is moved to and from the work on a slide actuated by a disk cam at the right-hand end of the machine. Two disk cams on the front feed shaft move the front and rear cross slides. Special cams are made for each different job.

The automatic screw machine of Fig. 21-8 is designed so that tools can act independently or together and noncutting movements can be overlapped to minimize operation time. A swing stop for locating stock leaves all six turret stations available for operating tools. The turret may be double indexed to save time where only two or three turret tools are needed. A large number of standard toolholders and other tools, similar to those on turret lathes, are available. A variety of standard and special attachments broaden the area of work that can be done. These include attachments for screw slotting, burring, turret drilling and tapping, thread chasing, cross drilling, and milling.

Various other single spindle automatics are available to handle work ranging from minute parts like watch staffs to bars 7¾ in. in diameter.

Automatic turret lathes. The automatic turret lathe in Fig. 21-9 has a headstock, turret, and cross slide on the bed, like all turret lathes. However, hand controls are absent, and the action of the

machine is controlled by drum cams in the base, one under the saddle and one beneath the cross slide. Speeds and feeds are changed mechanically for each step in an operation. An air-operated chuck is mounted on the spindle. Block-type toolposts are carried on the cross slide.

Multiple spindle automatics. Automatic screw and chucking machines are built with four, five, six, and eight spindles. Bar-type

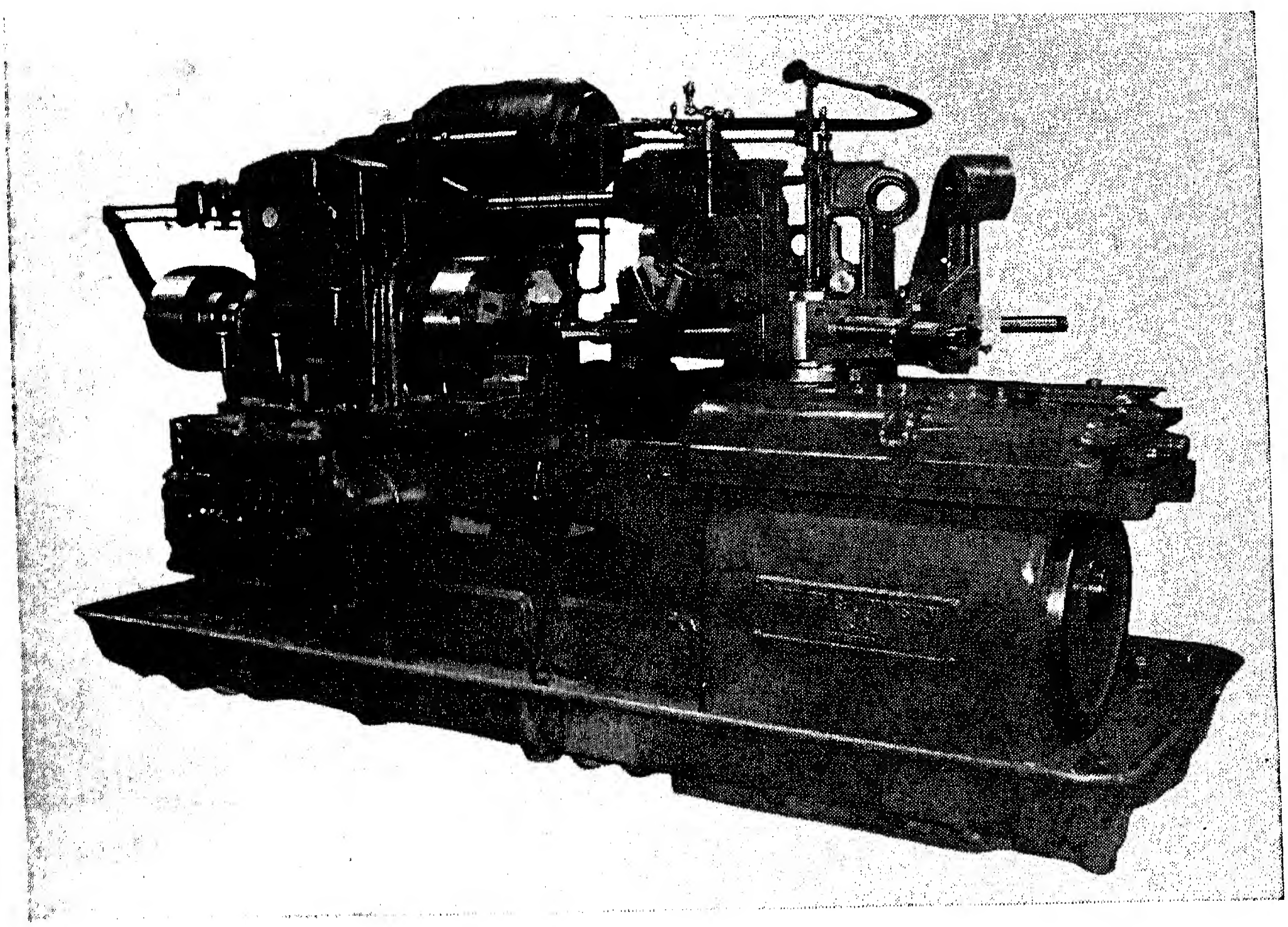


Fig. 21-9. An automatic turret lathe. (Courtesy Potter and Johnston Co.)

machines are rated by the largest diameter of stock that can be fed through the spindles. Some take bars as large as $2\frac{5}{8}$ in. in diameter. The capacity of a chucking machine is equivalent to the diameter of work that can be swung over the tool slides. The spindles are arranged on a circle in a large drum or spindle carrier on indexing-type machines. On another type used for simple work, the spindles are placed in a vertical row and do not index.

A typical six spindle automatic screw machine of the indexing type is shown in Fig. 21-10. Each work spindle carries a bar of

stock, revolves continuously, and is moved from station to station as the carrier indexes. The spindle carrier is indexed by a modified Geneva motion actuated from the main camshaft of the machine and is locked in position between indexing periods. A piece is machined in stages as it proceeds from station to station.

All the tools are brought to the work at the same time, instead of successively. The end working tools are carried on the main tool slide that has its axis in line with the axis of the spindle carrier.

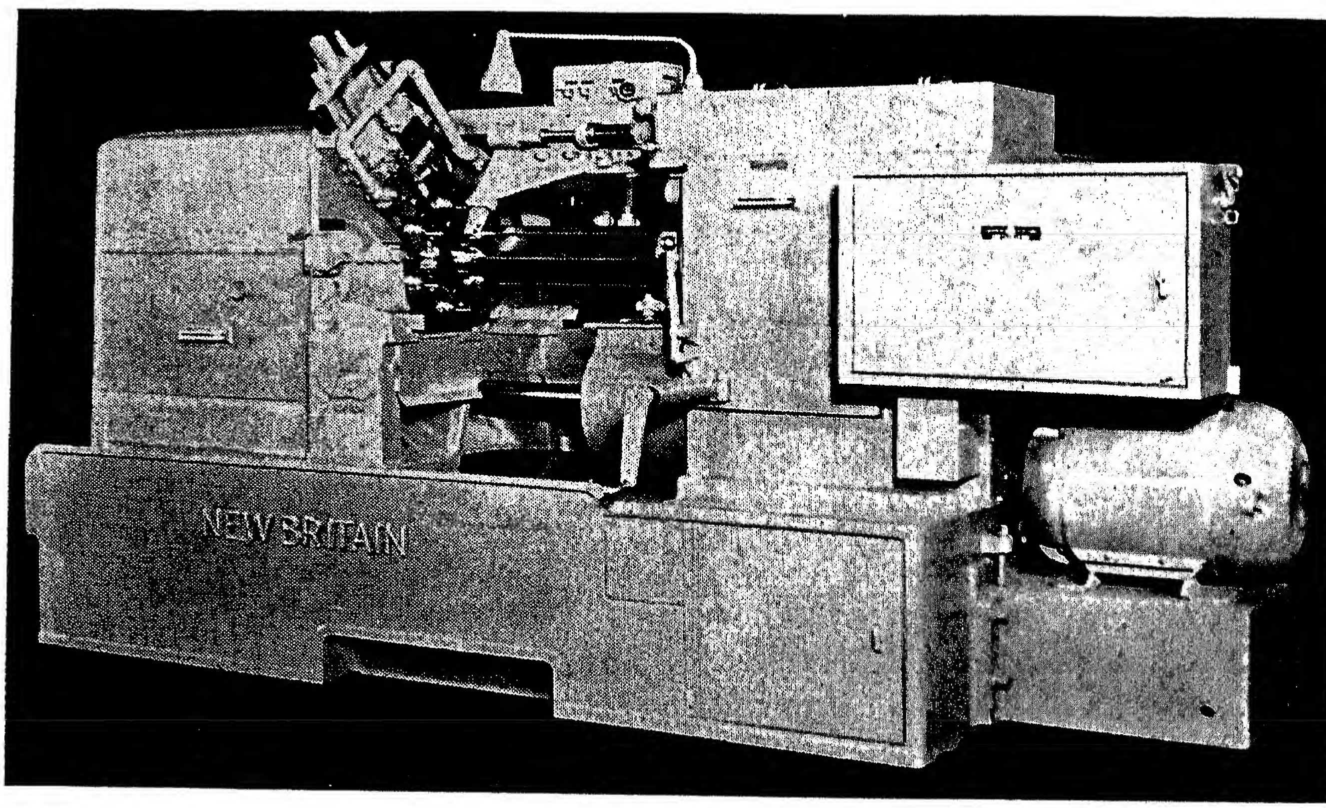


Fig. 21-10. A six spindle automatic screw machine. (Courtesy New Britain Machine Co.)

This slide does not turn but moves longitudinally to and from the carrier. On some machines the end working tools are carried on separate slides and can be fed at different rates. Sometimes, attachments are added to revolve drills, reamers, or taps to increase or decrease the relative speeds between the tools and work.

Side or cross slides for forming and cut-off tools are mounted next to the spindle stations and move radially to and from the center of the work. On some machines, cross slides have a longitudinal as well as radial motion with respect to the work for relieving, contour turning, and special grooving.

All spindles usually are indexed one position at a time, and a piece

is completed for each index. Some six and eight spindle machines are arranged for double indexing. On them, a spindle is indexed two positions each time, and two pieces are completed for each index. On a single indexing machine, a bar is fed forward or pieces are loaded only at one station. That function is performed at two stations on a double indexing machine.

A multiple spindle automatic actually operates on several pieces at one time. The time to make one piece on a single indexing machine is the time for the longest cut plus the indexing time. This time may be reduced by apportioning parts of long cuts to two or more spindles. For instance, a 2 in. length may be turned for a distance of one inch at one station and the rest of the way at a second station. The drilling of deep holes may be divided in the same way.

Comparison of single and multiple spindle automatics. A multiple spindle automatic is larger, more complicated, and costs more than a single spindle automatic for the same size of work. Thus a multiple spindle machine is economical only for those jobs on which it can turn out parts faster than a single spindle machine; that is, fast enough to make up at least for the extra fixed and overhead costs. In addition, more time is required to set up a multiple spindle automatic, and long runs are necessary to make up for the additional setup time. A rule of thumb is that a multiple spindle automatic is not justified for a job unless it can be kept busy for four or more days.

A multiple spindle automatic is faster than a single spindle automatic because all its tools work at once. However, under some circumstances a multiple spindle machine is little or no faster. The type of gear drive necessary on a multiple spindle automatic limits its spindle speed to less than that attainable on a single spindle automatic. That gives a single spindle machine an advantage for small pieces and soft materials like brass. Single spindle automatics can be indexed faster than multiple spindle machines. That is advantageous where cutting time is short as compared to idle time devoted to indexing and moving tools.

Speed and quantity of output are not the deciding factors under some circumstances. The single spindle automatic is considered more accurate than the multiple spindle machine which has more parts, each adding an error. The accuracy of a multiple spindle

automatic depends upon precise indexing, which is difficult to achieve. Also, single spindle automatics are available for larger work than can be accommodated on conventional multiple spindle automatics.

Questions

1. What are the two main types of horizontal turret lathes? What are their relative merits?
2. Describe a typical ram-type turret lathe.
3. Describe a typical saddle-type turret lathe.
4. Describe the typical toolholders and tools used on turret lathes.
5. What determines whether a job should be assigned to an engine lathe, turret lathe, or automatic?
6. Why are some pieces made from bar stock and others from forgings or castings?
7. How does the form of raw material affect the selection of a turret lathe and tooling?
8. What may be done in planning turret lathe operations to keep down the costs for setup, work handling, machine handling, and cutting?
9. What are multiple cuts and combined cuts? What advantages do they offer?
10. What is the difference between a fully automatic and a semiautomatic machine?
11. Describe a typical single spindle automatic screw machine.
12. Describe a typical multiple spindle automatic.
13. Discuss the relative merits of single spindle and multiple spindle automatics.

References

Longstreet, J. R. and Bailey, W. K., *Turret Lathe Operator's Manual.* Cleveland, Ohio: The Warner and Swasey Co., 1940.

Pattison, Donald M., "Automatic or Hand Operated," *Machinery*, May, 1950.

Manufacturers' Catalogs:
Bardons and Oliver, Inc., Cleveland, Ohio.
Brown and Sharpe Mfg. Co., Providence, R.I.
Cleveland Automatic Machine Co., Cincinnati, Ohio.
Cone Automatic Machine Co., Windsor, Vt.
Foster Division, International Machine Tool Corp., Elkhart, Ind.
Gisholt Machine Co., Madison, Wis.

George Gorton Machine Co., Racine, Wis.
Jones and Lamson Machine Co., Springfield, Vt.
National Acme Co., The, Cleveland, Ohio.
New Britain Machine Co., The, New Britain, Conn.
Potter and Johnston Co., Pawtucket, R.I.
Pratt and Whitney Division, Niles-Bement-Pond Co., W. Hartford, Conn.
Warner and Swasey Co., The, Cleveland, Ohio.

INDEX

4